Bricsys®

Command reference (V25)

Product documentation



Bricsys®

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1. Command reference

1.1 Executing Commands

In BricsCAD, you can launch commands in the following ways:

- Command Bar: by typing the CommandName
- Menu Bar: searching in the menu bar the CommandName, e.g. Draw | Line
- Ribbon: searching in the ribbon the CommandName, e.g. Home | Draw | Line
- Toolbar: searching in the toolbar the CommandName, e.g. Draw |

  Note: To open a specific toolbar, place the cursor on a toolbar, right-click and open **Toolbars** > **BricsCAD** and choose the toolbar that you want to be visible on the display.

- Quad tab: searching in the quad tab the CommandName, e.g. Draw |

1.2 Command options

Command options appear in the status bar or command bar and also in a prompt menu, depending on the PROMPTMENU system variable setting.

The PROMPTOPTIONFORMAT system variable controls how command options display in the command line prompts.

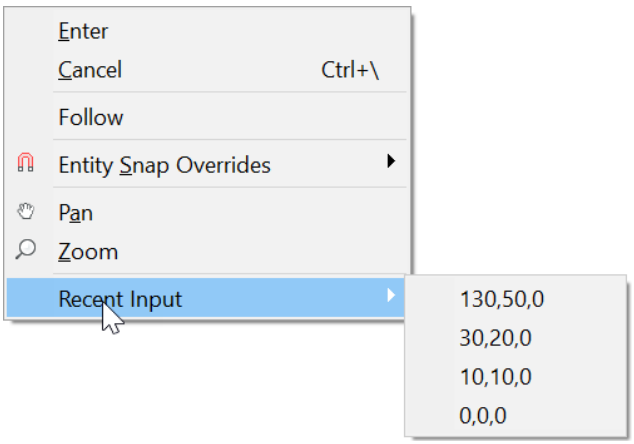
1.3 Accessing the input history

You can access and reuse the history of recent commands input in two ways:

1 **From the Command line**
Use the Up Arrow and Down Arrow keys at the Command prompt or at an input prompt.

Note: Markers for recent input of point locations are displayed in the drawing area. The markers are controlled by SNAPMARKERSIZE, SNAPMARKERCOLOR, and SNAPMARKERTHICKNESS system variables.

2 **From the command context menu**
Expand the **Recent Input** option in from the command context menu.



Note: The context menu can be accessed when the SHORTCUTMENU system variable is set to 4.

Remember: The input history can include the last 12 items from the current command (from the current and previous calls) and the last 12 items from other previous commands.

1.4 Command Prefixes

The following prefixes can be used with the BricsCAD commands:

Prefix	Type	Result
' (single quote or apostrophe)	Transparency	Allows a command to be executed inside another command. For example, if you are drawing a line, type **'zoom** at the command prompt to temporarily interrupt the LINE command while executing the ZOOM command.
- (dash)	Command line	Launches a non-dialog version of a command. All command options are available in the Command line. For example, type **-layer** in the command prompt to suppress the Drawing explorer / Layers dialog box. All command options are available in the Command line. This is especially useful if you need to execute such commands in a script.
_ (underscore)	Global	Executes the original English command (global command) in translated version of BricsCAD. It is recommended to use the global command prefix in scripts, or when defining keyboard shortcuts or customized tools.
. (period)	Non-redefined	Executes commands that have been temporarily removed from BricsCAD using the UNDEFINE command.

Command prefixes can be combined. For example, if the ZOOM command is temporarily undefined '.zoom executes the ZOOM command transparently.

2. +

2.1 +STRUCTUREPANEL command

Opens the **Structure tree configuration file** dialog box.

Shape Lite Pro Mechanical BIM

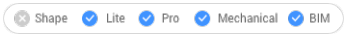
2.1.1 Description

Opens the **Structure tree configuration file** dialog box to select a cst file to include in the **Structure** panel. After choosing **Open** in the dialog box, the **Structure** panel displays.

3. ?

3.1 ? command

Opens the Bricsys Help Center.



3.1.1 Description

Opens the main page of the Bricsys Help Center website to learn more about BricsCAD commands, system variables, and workflows. It opens in your default web browser, allowing it to remain open while you work on your drawings in BricsCAD.

4. 2

4.1 2DINTERSECTION command

Toggles the intersection entity snap.

Shape Lite Pro Mechanical BIM

4.1.1 Description

Toggles the apparent intersection entity snap, which snaps to the intersections of entities, even when they only appear to be intersecting in 3D space.

Note: This command can be entered transparently during commands, but must be entered without the transparency apostrophe prefix (').

5. 3

5.1 3D command

Draws basic 3D mesh entities.

Shape Lite Pro Mechanical BIM

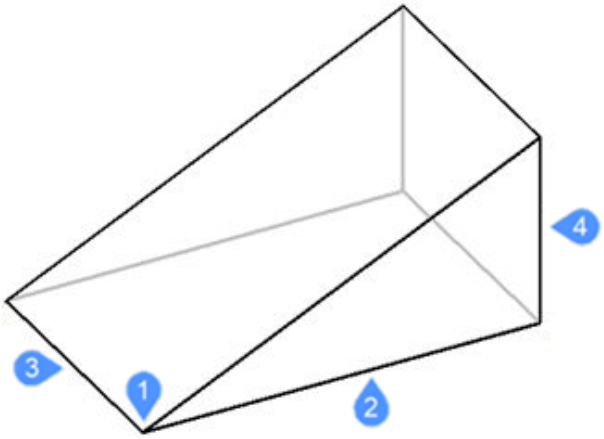
5.1.1 Description

Draws basic 3D mesh entities, including boxes, cones, cylinders, dishes, domes, meshes, pyramids, spheres, tori, and wedges.

5.1.2 Options within the command

Wedge

Draws 3D wedges (see AI_WEDGE command).



1 Corner of the wedge (Start point)

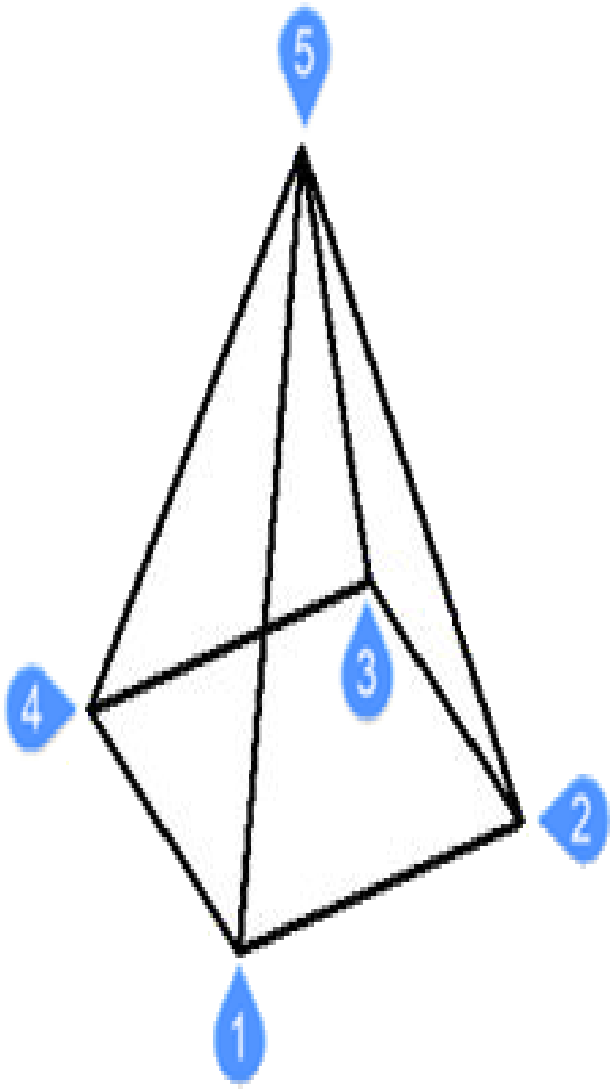
2 Length of the wedge

3 Width of the wedge

4 Height of the wedge

Pyramid

Draws 3D pyramids (see AI_PYRAMID command).



1 First point of base (Start point)

2 Second point

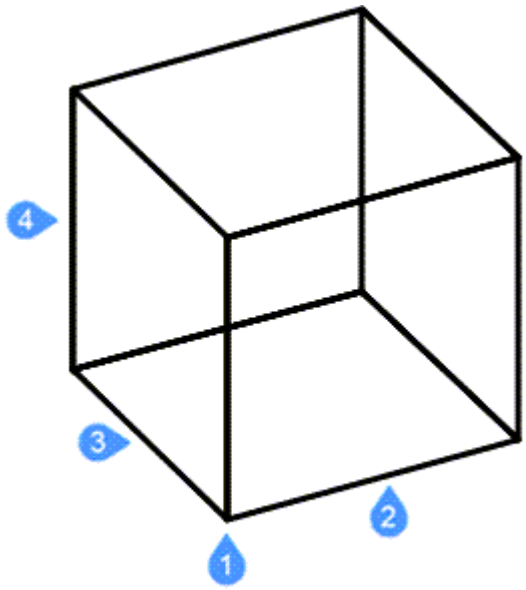
3 Third point

4 Last point for base

5 Set apex of pyramid

Box

Draws 3D boxes from polyface meshes (see AI_BOX command).



1 Corner of box (Start point)

2 Length of side of box

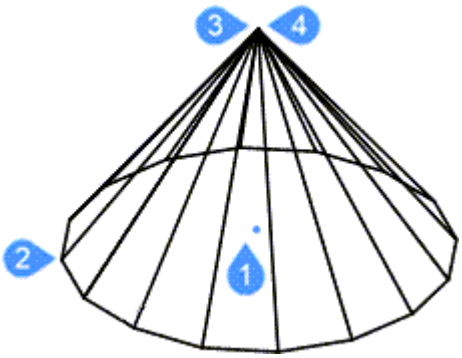
3 Width of box

4 Height of box

(Finish by applying rotation angle of box)

Cone

Draws 3D cones (see AI_CONE command).



1 Center of base of cone (Start point)

2 Radius of base of cone or Diameter

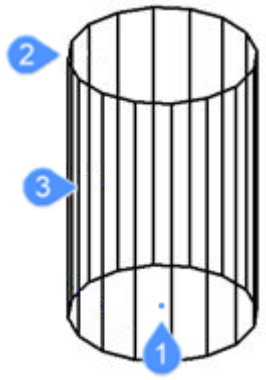
3 Radius of top of cone or Diameter

4 Height of cone

(Finish by applying number of segments in cone)

cyLinder

Draws 3D cylinders (see AI_CYLIDER command).



1 Center of base of cylinder (Start point)

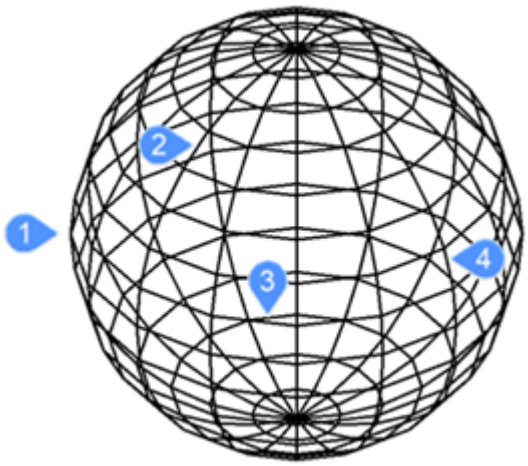
2 Radius of base of cylinder or Diameter

3 Height of cylinder

(Finish by applying number of segments in cylinder)

Sphere

Draws 3D spheres (see AI_SPHERE command).



1 Specify center of sphere (Start point)

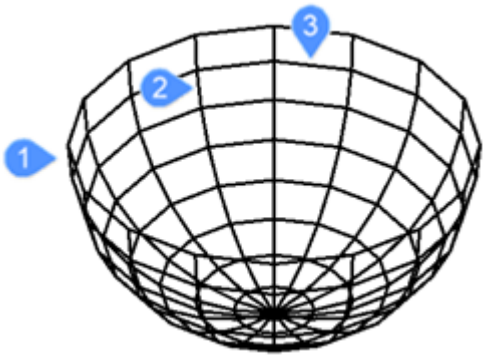
2 Radius of sphere (Diameter)

3 Number of longitudinal segments

4 Number of latitudinal segments

DIsh

Draws 3D dishes (bottom half of spheres) (see AI_DISH command).



Specify center of dish (Start point)

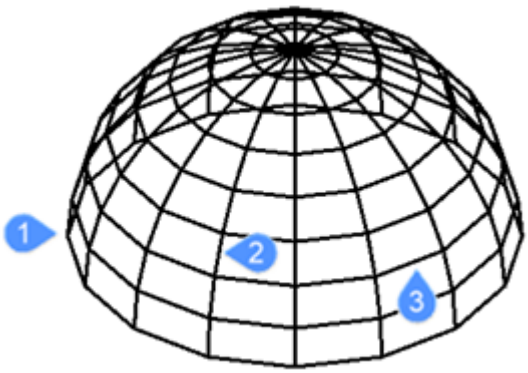
1 Radius of dish (Diameter)

2 Number of segments around dish

3 Number of top-to-bottom segments

DOme

Draws 3D domes (top half of spheres) (see AI_DOME command).



Specify center of dome (Start point)

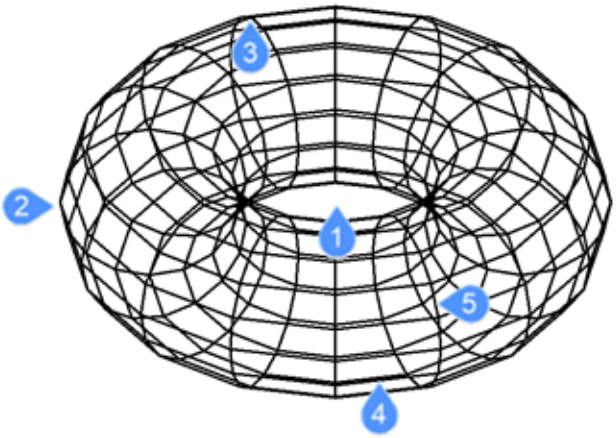
1 Radius of dome (Diameter)

2 Number of segments around dome

3 Number of top-to-bottom segments

Torus

Draws 3D tori (donuts) (see AI_TORUS command).



1 Specify center of whole torus (Start point)

2 Radius of whole torus

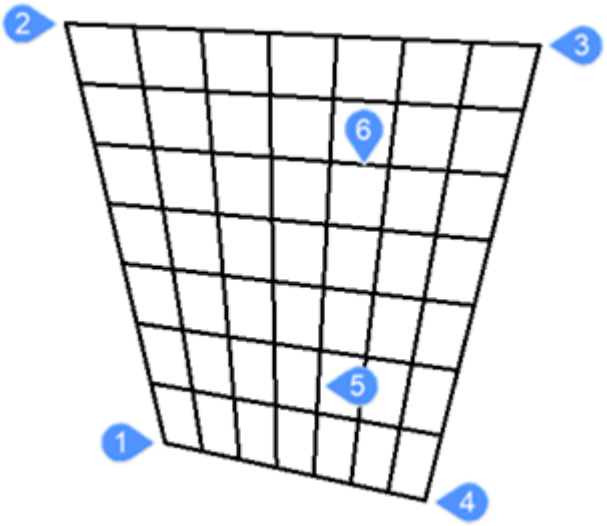
3 Radius of body of torus (Diameter)

4 Segments around body of torus

5 Segments around whole torus

Mesh

Draws 3D meshes (see 3DMESH command).



1 First of four corners of polygon mesh (Start point)

2 Second corner

3 Third corner

4 Last corner for mesh

5 Number of segments – primary axis

6 Number of segments – secondary axis

5.2 3DARRAY command

Constructs static rectangular and polar arrays in rows, columns, and levels.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: 3A, ARRAY3D

5.2.1 Description

Creates 3D rectangular arrays using rows, columns, and levels and 3D polar arrays using a center axis/ center point and rotation angle.

5.2.2 Method

There are two methods to create 3D arrays:

- Polar
- Rectangular

5.2.3 Options within the command

Polar
Creates 3D polar arrays.

enter to specify Angle between items
Allows you to specify the angle between items.

Rectangular
Creates 3D rectangular arrays.

5.3 3DCOMPARE command

Opens the **Compare two models** dialog box.

Shape Lite Pro Mechanical BIM

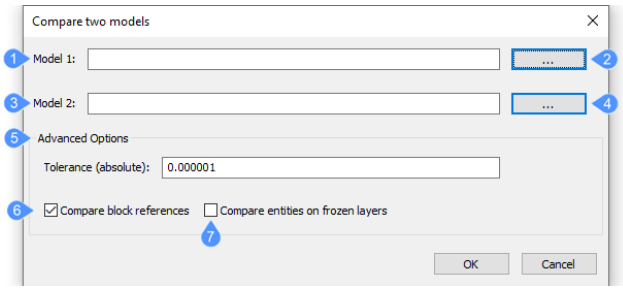
Icon: 

5.3.1 Description

Opens the **Compare two models** dialog box to compare solid and surface geometry between two drawing files.

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

5.3.2 Model 1

Specifies the file name of the first drawing to compare.

5.3.3 Select 1st drawing

Opens the Select first drawing for comparison dialog box to choose the drawing you want.

5.3.4 Model 2

Specifies the file name of the second drawing to compare.

5.3.5 Select 2nd drawing

Opens the **Select second drawing for comparison** dialog box to choose the drawing you want.

5.3.6 Advanced Options

The tolerance (absolute) expresses the threshold value for geometry comparison: determines how close two models must be to be considered unchanged.

5.3.7 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.

5.3.8 Compare entities on frozen layers

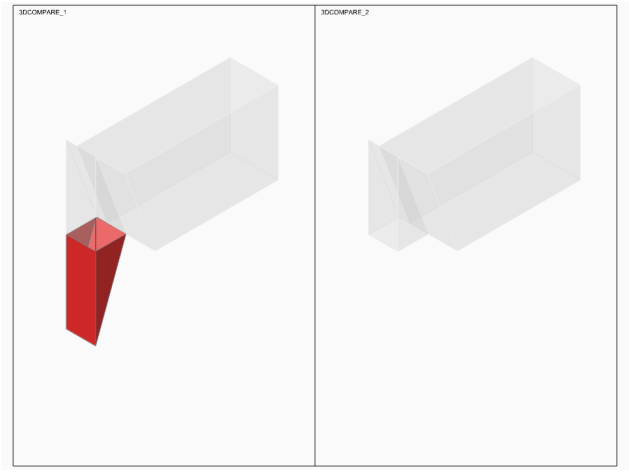
Determines if entities on frozen layers should be compared:

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

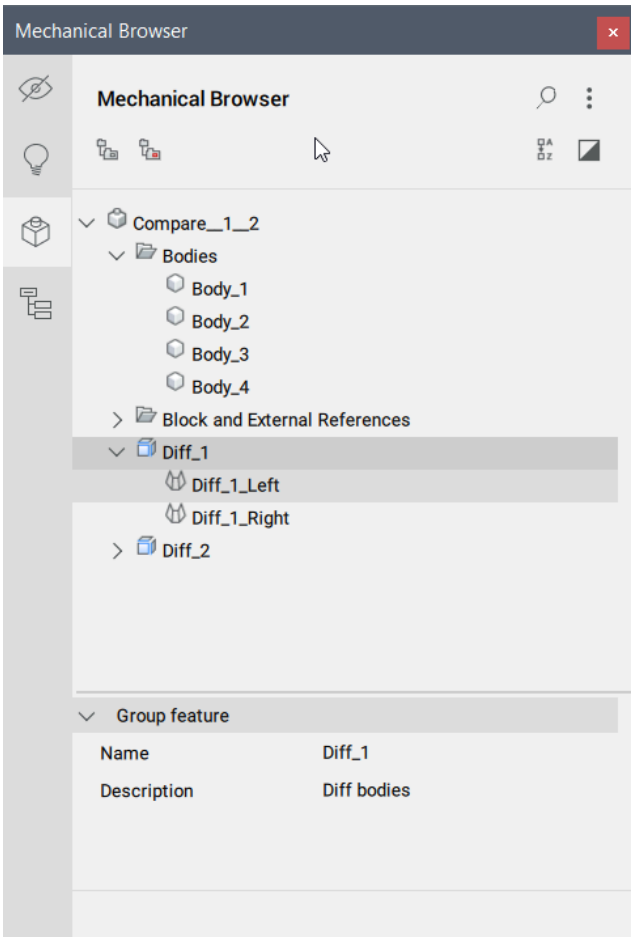
The value of the 3DCOMPAREMODE system variable controls the display in the viewports of the Comparison layout; in the **Settings** dialog box, search for "3dcomparemode".

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 two drawings.



5.4 3DCONVERT command

Converts 3D solids to polyface meshes.

Shape Lite Pro Mechanical BIM

5.4.1 Description

Converts 3D ACIS selected entities to 3D polyface meshes.

5.5 3DDWF command

Opens the **Export 3D DWF** dialog box.

Shape Lite Pro Mechanical BIM

5.5.1 Description

Opens the **Export 3D DWF** dialog box to save data from the current drawing to a 3D DWF or DWFX file. The default file format is specified by the DWFFORMAT system variable.

5.6 3DFACE command

Draws 3D 4-edged faces.

Shape Lite Pro Mechanical BIM

Icon:

Alias: 3F, FACE

5.6.1 Description

Draws 3D 4-edged faces with optional invisible edges, by entering values in the Command line or picking points in drawing space.

5.6.2 Options within the command

select Invisible edge
Makes the next edge invisible.

Note: The edge appears invisible in the wireframe visual style.

5.7 3DINTERSECTION command

Toggles the intersection entity snap.

Shape Lite Pro Mechanical BIM

Icon:

5.7.1 Description

This command can be entered transparently during commands, but must be entered without the transparency apostrophe prefix (').

Note: This entity snaps to entities that physically intersect each other. To snap to apparent intersections, use 2DINTERSECTION entity snap.

5.8 3DMESH command

Creates 3D surface meshes.

Shape Lite Pro Mechanical BIM

Icon:

Alias: MESH

5.8.1 Description

Creates 3D surface meshes by entering the number of vertices and their position.

Note: The value of the FACETRES system variable influences the display resolution of a mesh.

5.8.2 Options within the command

Number of vertices in M direction (between 2 and 256)
Specifies the number of vertices in the primary direction.

Number of vertices in N direction (between 2 and 256)
Specifies the number of vertices in the secondary direction.

Position of vertex (M,N)
Specifies the position of the vertex.

Note: 3D meshes can be edited directly through grips.

5.9 3DOSNAP command

Opens the **Settings** dialog box with the **Entity 3D snap mode** category expanded.

Shape Lite Pro Mechanical BIM

Icons: 

5.9.1 Description

Opens the **Settings** dialog box with the **3D entity snap mode** category expanded to view and modify relevant system variables.

5.10 -3DOSNAP command

Toggles entity snap modes for 3D entities.

Shape Lite Pro Mechanical BIM

5.10.1 Description

Toggles 3D entity snap modes for drawing and editing entities in 3D space at the Command line (short for "3D object snap").

5.10.2 Options within the command

nearest (ZNEA)
Snaps to a point on the face of a 3D entity that is nearest to the cursor, equivalent to the ZNEAREST command.

vertex (ZVER)
Snaps to the closest vertex of a 3D entity, equivalent to the ZVERTEX command.

midpoint (ZMID)
Snaps to the midpoint of a face's edge, equivalent to the ZMIDPOINT command.

center (ZCEN)
Snaps to the center of a flat or curved 3D face, equivalent to the ZCENTER command.

perpendicular (ZPER)
Snaps to a point perpendicular to a face, equivalent to the ZPERPENDICULAR command.

Knot (ZKNO)
Snaps to a knot on a spline, equivalent to the ZKNOT command.

intersection (ZINT)
Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces, equivalent to the ZINTERSECTION command.

PointCloud Nearest
Snaps to the nearest point cloud point.

NON
Turns off all 3D snap modes, equivalent to the ZNONE command.

ON
Enables all currently set 3D snap modes.

OFF
Disables all currently set 3D snap modes.

5.11 3DPOLY command

Creates a 3D polyline.

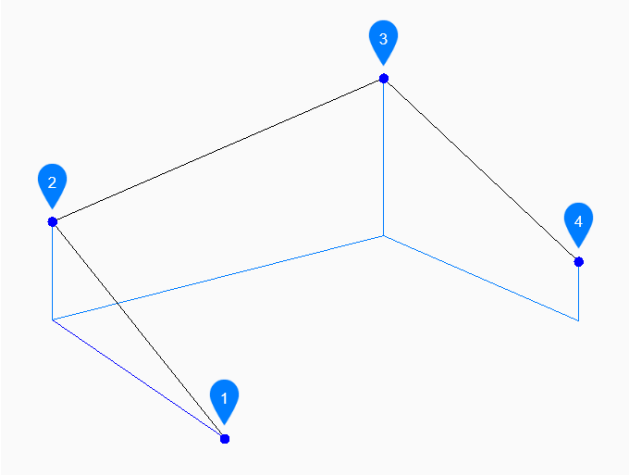
Shape Lite Pro Mechanical BIM

Icon:

Alias: 3P

5.11.1 Description

Creates a single 3D polyline entity with multiple line segments by specifying the start and end point of each segment. Options allow you to undo and close the geometry.



1 Start

2 Next

3 Next

4 End

5.11.2 Method

This command has 3 methods to begin creating a 3D polyline:

- Start of polyline
- Last point
- Follow

You can continue adding unlimited segments until you press **Enter** to end the command.

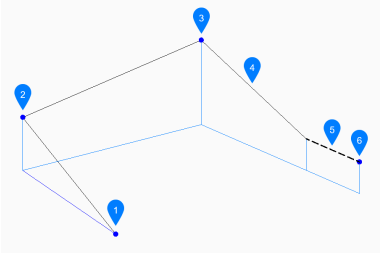
Note: 3D polylines with assigned linetypes draw with an uninterrupted pattern. To make linetypes visible for 3D polylines, set LINETYPE3DPLINE system variable to 1.

5.11.3 Options within the command

Start of polyline
Begins creating a 3D polyline by specifying a start point then specify next point.

Last point
Begins creating a 3D polyline from the last point picked then specify next point.

Follow
Begins creating a 3D polyline from the last drawn arc or line segment, following its angle, then specify next point.



1 Start

2 Next

3 Next

4 Last Drawn Segment

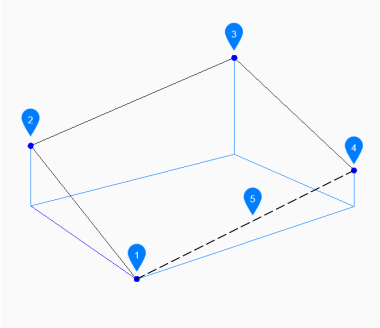
5 Follow

6 End

Next point
Specifies the next vertex of the polyline.
For the **Follow** option, specifies the length of the segment. Picking a point only determines the length since the angle follows the last segment.

Undo
Undoes the last 3D polyline segment and continue drawing from its previous start point.

Close
Automatically draws a 3D polyline segment from the end point of the last segment to the start point of the first segment. This ends the command.



1 Start/End

2 Next

3 Next

4 Next

5 Last drawn segment

5.12 3DROTATE command

Rotates entities around an axis in 3D space.

Shape Lite Pro Mechanical BIM

5.12.1 Description

Dynamically rotates 3D solids, surfaces, 2D entities, faces (planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface), edges, or vertices of a solid around an axis.

Note: When the face of a solid or surface is rotated, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.

5.12.2 Method

There are two methods to rotate entities:

- Specify an axis.
- Use the geometry of an object as the axis for rotating the selected entities.
  - Hover over a line or linear polyline segment. Click when the entity highlights.
  - Hover over a 3D solid face, then move the cursor close to the edge you want to use as the rotation axis. A rotation arc and the rotation axis display dynamically. Click to accept the rotation axis.

5.12.3 Options within the command

Xaxis

Rotates the selection around the X-axis of the current coordinate system.

Yaxis

Rotates the selection around the Y-axis of the current coordinate system.

Zaxis
Rotates the selection around the Z-axis of the current coordinate system.

2Points
Defines the rotation axis by two points.

Object
Uses the geometry of an object to determine the axis for rotating the selected entities.
Click to start the rotation of the selected entity. It automatically selects a rotation axis if the cursor is over a planar face.

Last
Uses the last entered axis for rotating the entities.

View
Uses the current view as the axis for rotating the entities.

Note: The axis of rotation is parallel to the direction of view, passing through the picked point.

Rotation angle
Specifies the angle by which to rotate the entities.

Base angle
Allows you to define a new base angle.

Copy
Rotates a copy of the selection set.

Repeat
Creates multiple copies.

Disable connectivity mode
Removes the connectivity with adjacent entities.

Enable connectivity mode
Keeps the connectivity with adjacent entities.

Note: If the HOTKEYASSISTANT system variable is on, press the Ctrl key to cycle between disable/enable connectivity mode.

Note: Geometric and dimensional constraints between selected entities are taken in consideration, as well as the lower and upper bounds parameters for 3D dimensional constraints. Constraints can be accessed and modified through the **Mechanical Browser** panel.

6. A

6.1 ABOUT command

Opens the **About BricsCAD** dialog box.

Shape Lite Pro Mechanical BIM

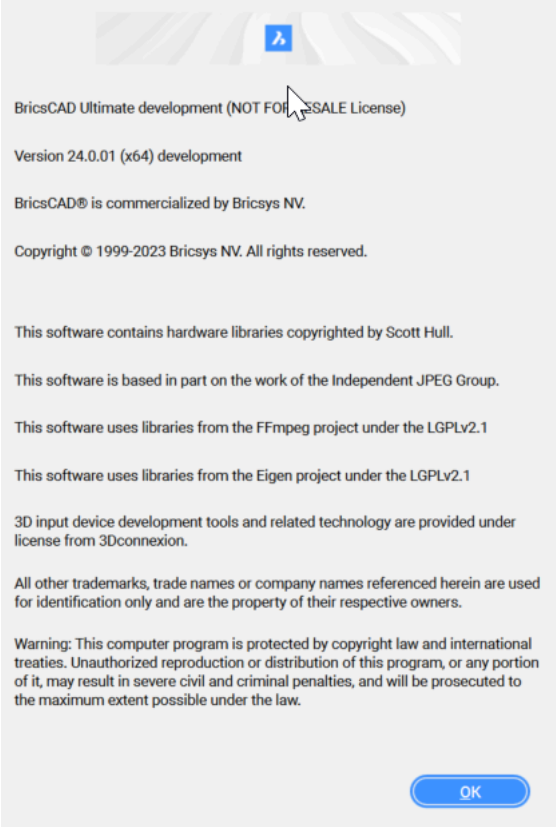
Icon: ⓘ

6.1.1 Description

Opens the **About BricsCAD** dialog box.

The **About BricsCAD** dialog box displays BricsCAD version, copyright and product information.

The product information includes the license type, version and revision number.



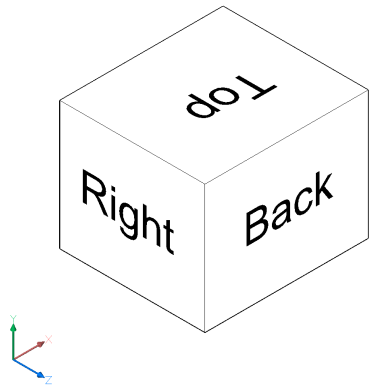
6.2 ACETUCS-BACK command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the back plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.2.1 Method

The UCS automatically changes when running the command.



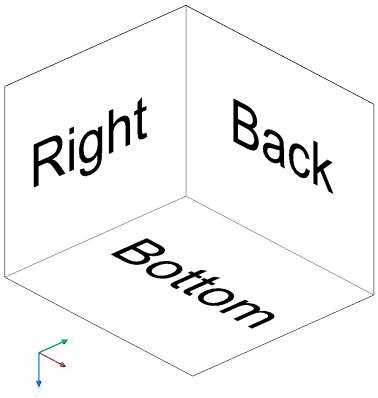
6.3 ACETUCS-BOTTOM command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the bottom plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.3.1 Method

The UCS automatically changes when running the command.



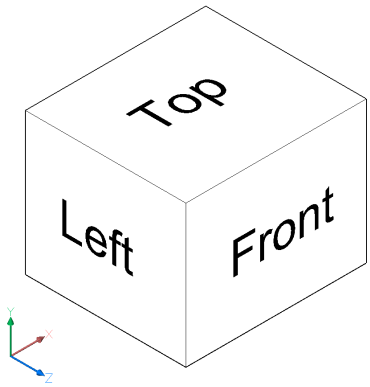
6.4 ACETUCS-FRONT command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the front plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.4.1 Method

The UCS automatically changes when running the command.



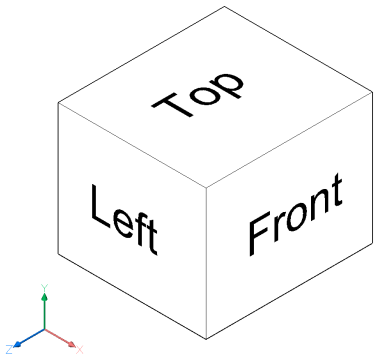
6.5 ACETUCS-LEFT command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the left plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.5.1 Method

The UCS automatically changes when running the command.



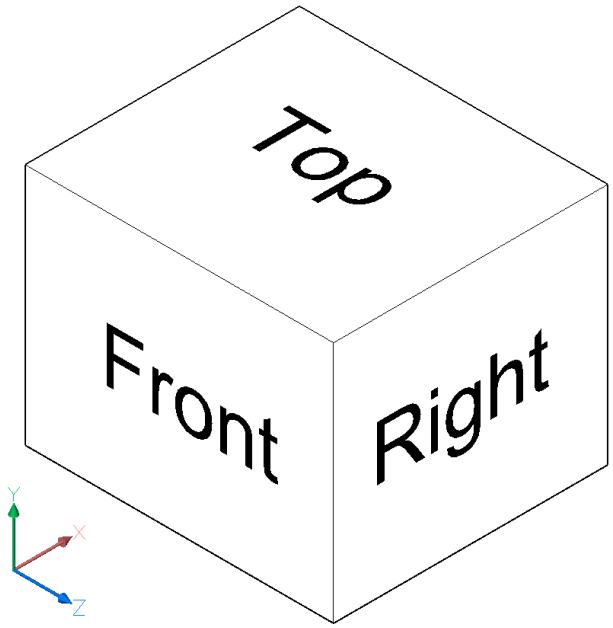
6.6 ACETUCS-RIGHT command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the right plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.6.1 Method

The UCS automatically changes when running the command.



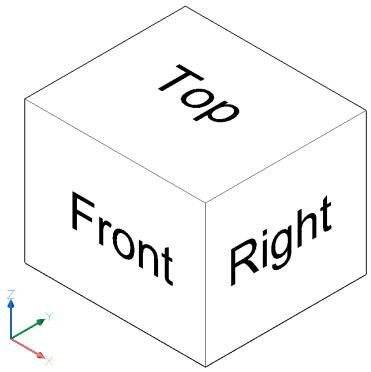
6.7 ACETUCS-TOP command (Express Tools)

Changes the UCS so that the XY-plane is parallel to the top plane of the drawing and the Z-axis points away from it.

Shape Lite Pro Mechanical BIM

6.7.1 Method

The UCS automatically changes when running the command.



6.8 ACISIN command

Imports .sat or .sab files into the current drawing.

Shape Lite Pro Mechanical BIM

Icon:

6.8.1 Description

Opens the **Open an ACIS file** dialog box to select a .sat or .sab file to import into the current drawing.

Note: The ACISIN command ignores the INSUNITS system variables of the .sat and .dwg files, so one unit of the imported SAT or SAB file is always converted to one DWG drawing unit.

6.9 ACISOUT command

Exports solid and surface models in ACIS format for use by other solid modeling programs.

Shape Lite Pro Mechanical BIM

Icon:

6.9.1 Description

Opens the **Create an ACIS file** dialog box to export 3D solids and surface entities in ACIS format, either in human-readable ASCII .sat or compact binary .sab format. Exported files can be shared with other 3D CAD programs capable of solid modeling or analyzing solid models.

Note: The scale factor stored in the file is always 1.

6.10 ACTIVATESECTIONVIEW command

Sets viewpoint by section plane.

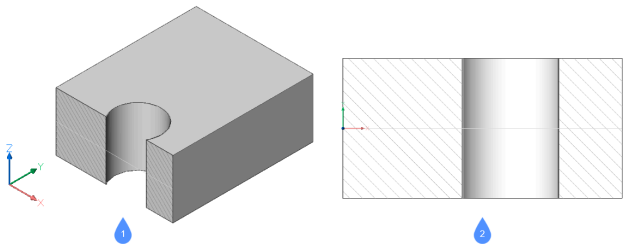
Shape Lite Pro Mechanical BIM

6.10.1 Description

Note: It requires at least one section plane in the drawing, made with the SECTIONPLANE command.

The command:

- reorients the viewpoint to be head-on to the selected section plane.
- creates a UCS in the selected section plane.
- sets Off the **Perspective** view.



1 Section plane

2 The result of the command for the section plane.

6.11 ACTIVATESTORY command

Activates a named story.

Shape Lite Pro Mechanical BIM

6.11.1 Method

Enter the building and story numbers.

Note: Building and story numbers are listed in the Command line along with their names.

Decide whether or not to show the story in **Top View Mode**. The default value is **Yes**.

6.11.2 Options within the command

Yes
Activates the story in **Top View Mode**.

No
Activates the story without any view modification.

6.12 ADDINMAN command

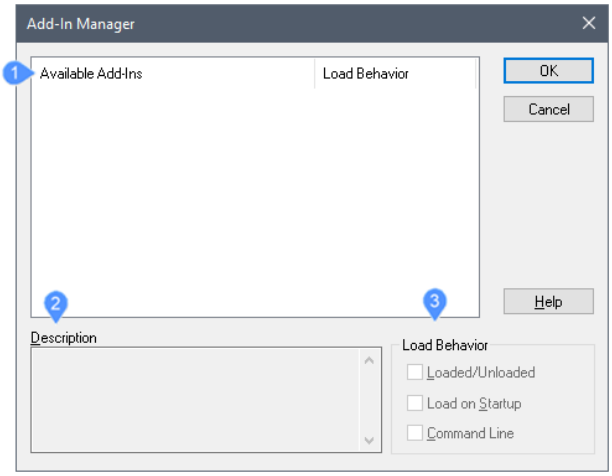
Opens the **Add-in Manager** dialog box.

Shape Lite Pro Mechanical BIM

6.12.1 Description

Opens the **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

6.12.2 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.

6.12.3 Description

Describes the selected VBA COM module, if any.

6.12.4 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.

6.13 ADDLINECURVELABELS command

Adds labels along curves or lines.

Shape Lite Pro Mechanical BIM

Icon: 

6.13.1 Method

Specify the general label style names for lines and curves and select the entities (lines, polylines 3D polylines,Civil String entities) to which the labels apply. The available label style names for lines and curves are listed in the Command line.

Note: You can create or edit the label style for lines and curves by accessing the **Label Style Editor** dialog box from **Civil Explorer** panel - **Settings** tab.

The line and curve label has two grips. One moves the label along the segment and the other drags it away from the segment.

Note: You can edit the dragged state of the label style by accessing the Dragged state tab in **Label Style Editor** dialog box from **Civil Explorer** panel - **Settings** tab.

6.14 ADDSELECTED command

Creates an entity similar to an existing one.

Shape Lite Pro Mechanical BIM

Icon: 

6.14.1 Method

Begin adding a new entity to the drawing by selecting an existing entity of the same type and properties you wish to add. BricsCAD automatically launches the appropriate command and applies properties to create a matching entity. For instance, select a spline and the program immediately starts the Spline command. Some entities are not yet supported.

6.15 ADDSURFACELABELS command

Creates surface labels.

Shape Lite Pro Mechanical BIM

Icon: 

6.15.1 Description

Creates spot elevation levels, slope labels or contour labels on a surface.

6.15.2 Method

Choose the type of the surface label and select the surface to apply to.

6.15.3 Options within the command

Contour
Creates major and minor contour labels using the available label styles.

spot Elevation
Creates spot elevation labels using an available label style with a specified symbol style name.

Grid
Places spots elevation on a grid with a specified X and Y spacing, and rotation.

Slope
Creates slope labels using the available label styles.

Single point
Creates a slope label by indicating a point on a surface.

Two points
Creates a slope label by indicating two points.

6.16 AI_BOX command

Creates a 3D polyface mesh in the shape of a box.

Shape Lite Pro Mechanical BIM

Icon: 

6.16.1 Description

Creates a 3D polyface mesh in the shape of a rectangular or square box. Choose from a combination of options including corner, length, width, height, cube, and rotation angle.

6.16.2 Method

There is one method to begin creating a box:

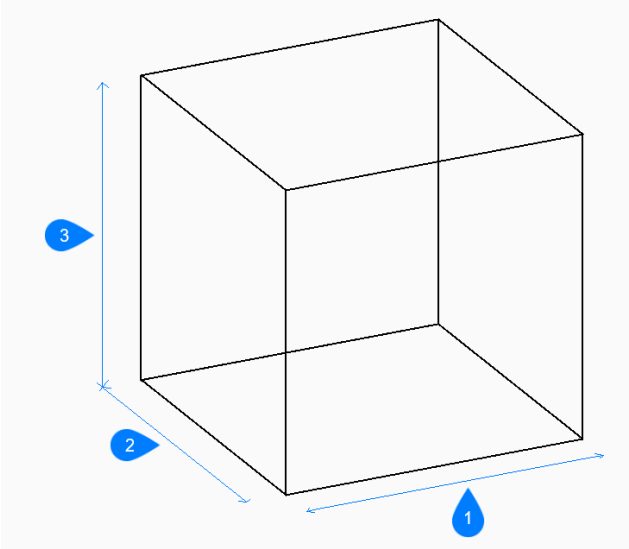
- Corner of box

6.16.3 Options within the command

Length of side of box
Specifies the length of the box.

Set width of box
Specifies the width of the box.

Cube
Specifies a single distance to use for the length, width, and height of the box.



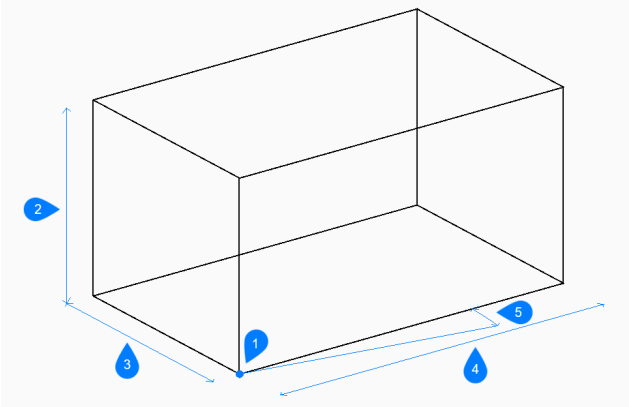
1 Length

2 Width

3 Height

Height of box
Specifies the height of the box.

Rotation angle for box
Specifies the rotation angle of the base of the box in the xy-plane. The default value is 0.



1 Corner of box

2 Height

3 Width

4 Length

5 Rotation angle

6.17 AI_CONE command

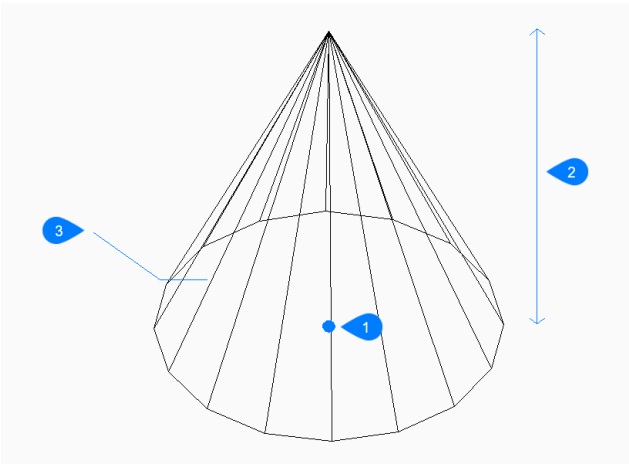
Creates a 3D polygon mesh in the shape of a cone.

Shape Lite Pro Mechanical BIM

Icon:

6.17.1 Description

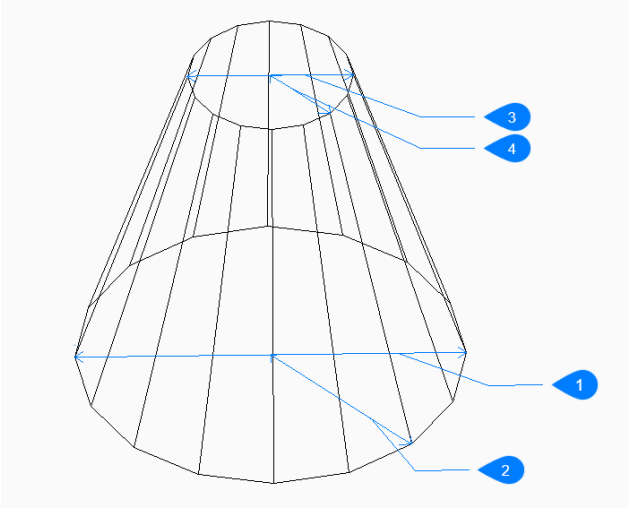
Creates a 3D polygon mesh in the shape of a circular or elliptical cone. Choose from a combination of options including center, radius, diameter, height, axis endpoints, apex, and number of segments.



1 Center

2 Height

3 Segments

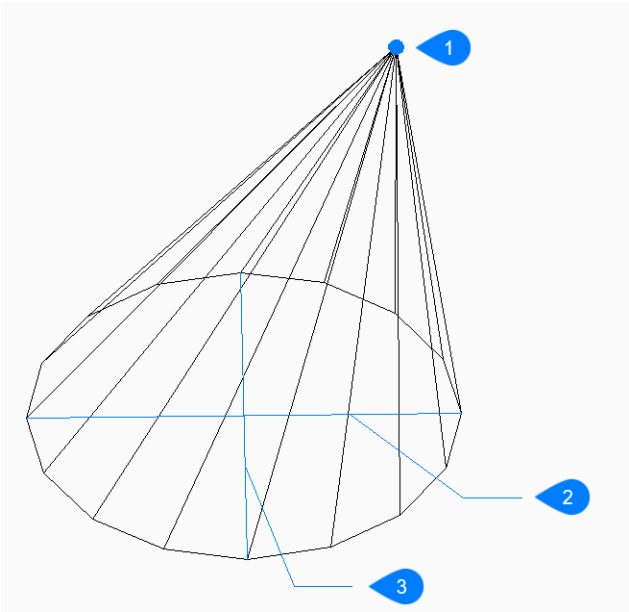


1 Base diameter

2 Base radius

3 Top diameter

4 Top radius



1 Apex

2 Ellipse axis

3 Other axis

6.17.2 Methods

This command has 2 methods to begin creating a cone:

- Center of base of cone
- Elliptical

6.17.3 Options within the command

Center of base of cone
Specifies the center of the base of the cone.

Radius of base of cone
Specifies the radius of the base of the cone.

Diameter
Specifies the diameter of the base of the cone.

Radius of top of cone
Specifies the radius of the top of the cone.

Diameter of top of cone
Specifies the diameter of the top of the cone.

Elliptical
Allows you to create an elliptical cone by specifying the first end of the ellipse axis.

Center
Specifies the center of elliptical base.

Endpoint of axis
Specifies the endpoint of axis.

Second end of ellipse axis
Specify the second end of the ellipse axis.

Other axis
Specify the radius of the other axis.

Apex
Specify the location for the top of the cone.

Height of cone
Specifies the height of the cone.

Number of segments
Specifies the number of isolines used to simulate the cone's curved surface. The default value is 16.

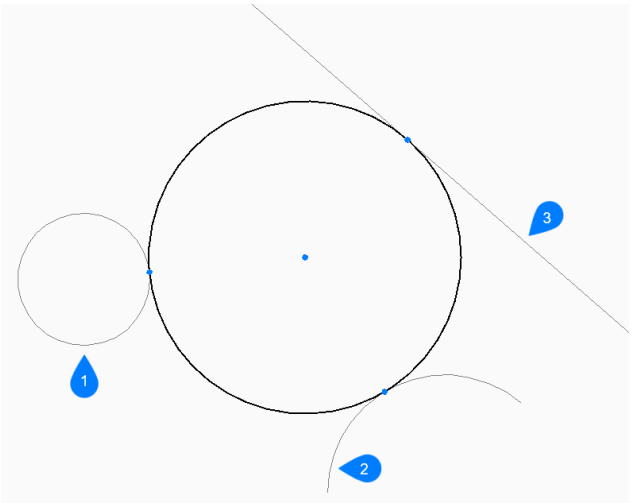
6.18 AI_CIRCTAN command (Express Tools)

Creates a tangent circle.

Shape Lite Pro Mechanical BIM

6.18.1 Description

Creates a circle tangent to three entities that can include lines, polyline segments, polyarcs, circles, arcs, xlines or rays.



1 First tangent definition point

2 Second tangent definition point

3 Third tangent definition point

6.18.2 Method

Enter first, second and third tangent definition points.

6.18.3 Options within the command

First Tangent definition point
Begin creating a circle by selecting a tangent point on the first entity.

Second Tangent definition point
Allows you to select a tangent point on the second entity.

Third Tangent definition point
Allows you to select a tangent point on the third entity.

6.19 AI_CYLINDER command

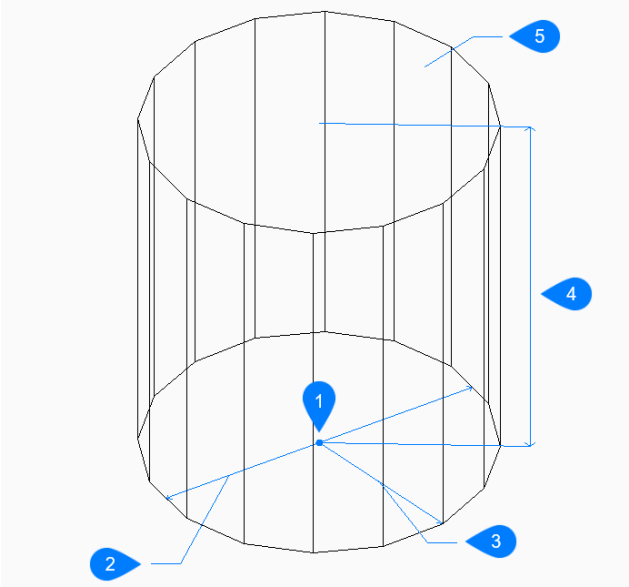
Creates a 3D polygon mesh in the shape of cylinder.

Shape Lite Pro Mechanical BIM

Icon:

6.19.1 Description

Creates a 3D polygon mesh in the shape of a circular or elliptical cylinder. Choose from a combination of options including center, radius, diameter, height, axis endpoints, and number of segments.



1 Center

2 Diameter

3 Radius

4 Height

5 Segment

6.19.2 Method

This command has 2 methods to begin creating a cylinder:

- Center of base of cylinder
- Elliptical

6.19.3 Options within the command

Center of base of cylinder
Specifies the center of the base of cylinder.

Radius of base of cylinder
Specifies the radius of base of cylinder.

Diameter
Specifies the diameter of base of cylinder.

Elliptical
Specifies the first end of the ellipse axis.

Second end of ellipse axis
Specifies the second end of the ellipse axis.

Center
Specifies the center of the ellipse.

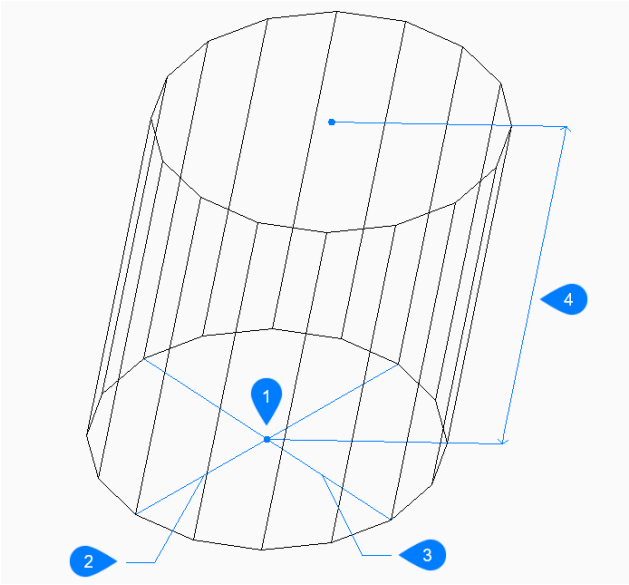
Endpoint of axis
Specifies the endpoint of the ellipse's axis.

Other axis
Specifies the radius of the other axis.

Center of second end
Specifies the center for the other end of the axis to create an oblique cylinder.

Height of cylinder
Specifies the height of the cylinder.

Number of segments
Specifies the number of isolines used to simulate the cylinder’s curved surface. The default value is 16.



1 Center

2 First axis

3 Other axis

4 Height

6.20 AI_DESELECT command (Express Tools)

Deselects all entities.

Shape Lite Pro Mechanical BIM

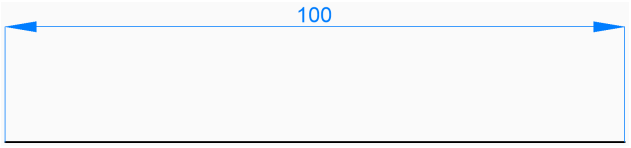
6.21 AI_DIM_TEXTABOVE command (Express Tools)

Places the DIM text above the dimension line.

Shape Lite Pro Mechanical BIM

6.21.1 Method

Select the dimension entities. The DIM text is automatically placed above the dimension line.



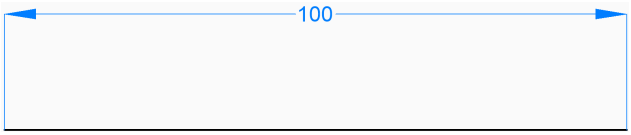
6.22 AI_DIM_TEXTCENTER command (Express Tools)

Places the DIM text in the center of the dimension line.

Shape Lite Pro Mechanical BIM

6.22.1 Method

Select the dimension entities. The DIM text is automatically placed in the center of the dimension line.



6.23 AI_DIM_TEXTHOME command (Express Tools)

Restores dimension texts to the default position.

Shape Lite Pro Mechanical BIM

Note: The command changes the dimension text position horizontally only.

6.24 AI_DISH command

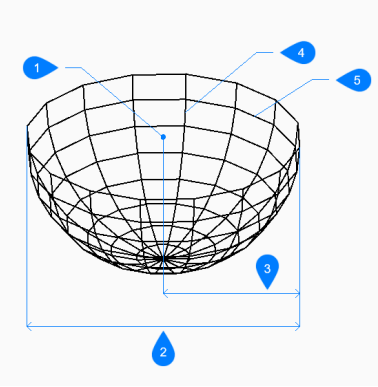
Creates a 3D polygon mesh in the shape of the bottom half of a sphere.

Shape Lite Pro Mechanical BIM

Icon:

6.24.1 Description

Creates a 3D polygon mesh in the shape of the bottom half of a sphere. Choose from a combination of options including center, radius, diameter and number of segments.



1 Center

2 Diameter

3 Radius

4 Top to bottom segments

5 Segments around dish

6.24.2 Method

There is one method to begin creating a dish:

- Center of dish

6.24.3 Options within the command

Center of dish
Specifies the center of the dish's open end.

Set radius of dish
Specifies the radius of the dish.

Diameter
Specifies the diameter of the dish.

Number of segments around dish
Specifies the number of isolines, perpendicular to the xy-plane, used to simulate the dish's curved surface. The default value is 16.

Number of top-to-bottom segments
Specifies the number of isolines, parallel to the xy-plane, used to simulate the dish's curved surface. The default value is 8.

6.25 AI_DOME command

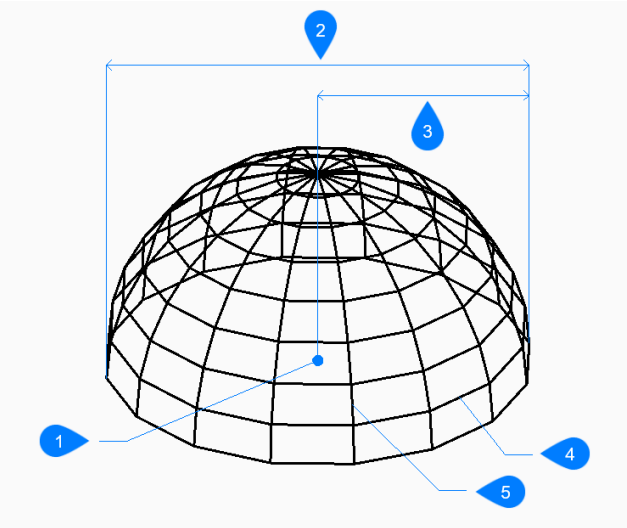
Creates a 3D polygon mesh in the shape of the upper half of a sphere.

Shape Lite Pro Mechanical BIM

Icon:

6.25.1 Description

Creates a 3D polygon mesh in the shape of the upper half of a sphere. Choose from a combination of options including center, radius, diameter and number of segments.



1 Center

2 Diameter

3 Radius

4 Top to bottom segments

5 Segments around dome

6.25.2 Method

There is one method to begin creating a dome:

- Center of dome

6.25.3 Options within the command

Center of dome
Specifies the center of the dome's open end.

Set radius of dome
Specifies the radius of the dome.

Diameter
Specifies the diameter of the dome.

Number of segments around dome
Specifies the number of isolines, perpendicular to the xy-plane, used to simulate the dome's curved surface. The default value is 16.

Number of top-to-bottom segments
Specify the number of isolines, parallel to the xy-plane, used to simulate the dome's curved surface. The default value is 8.

6.26 AI_DRAWORDER command (Express Tools)

Changes the display order of overlapping entities.

Shape Lite Pro Mechanical BIM

6.26.1 Method

Moves the visual display of selected entities:

- Above
- Under
- Front
- Back

6.26.2 Options within the command

Above
Moves the visual display of the selected entities above a reference entity.

Under
Moves the visual display of the selected entities under a reference entity.

Front
Moves the selected entities' display order above all other overlapping entities.

Back
Moves the selected entities' display order below all other overlapping entities.

6.27 AI_EDGESURF command

This command is obsolete. It still exists for backward compatibility only.

Shape Lite Pro Mechanical BIM

Use the EDGESURF command instead.

6.28 AI_FMS command (Express Tools)

Switches from paper space to model space viewports.

Shape Lite Pro Mechanical BIM

6.28.1 Method

If you are in model space, it switches to the last opened layout in paper space.

6.29 AI_MOLC command (Express Tools)

Sets current the layer of the selected entity.

Shape Lite Pro Mechanical BIM

6.30 AI_MSPACE command (Express Tools)

Switches to model space.

Shape Lite Pro Mechanical BIM

6.31 AI_PROPCHK command (Express Tools)

Opens the **Properties** panel for the selected entities.

Shape Lite Pro Mechanical BIM

6.32 AI_PSPACE command (Express Tools)

Switches to the last opened layout in paper space.

Shape Lite Pro Mechanical BIM

6.33 AI_PYRAMID command

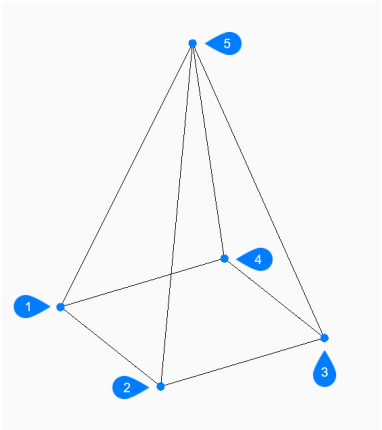
Creates a 3D polyface mesh in the shape of a pyramid.

Shape Lite Pro Mechanical BIM

Icon:

6.33.1 Description

Creates a 3D polyface mesh in the shape of a pyramid with three or four sides. The sides of the base may have unequal lengths. Options allow you to specify a pointed, ridged, or flat top.



1 First point

2 Second point

3 Third point

4 Last point

5 Apex

6.33.2 Methods

There is one method to begin creating a pyramid

- First point for the base of pyramid

6.33.3 Options within the command

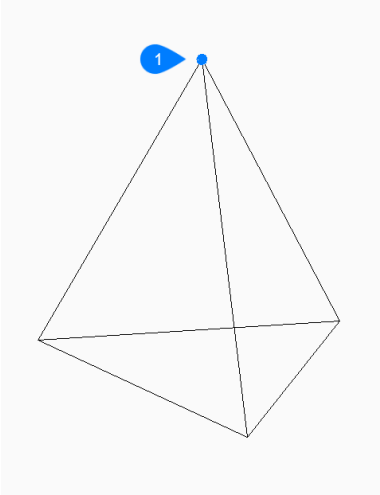
First point for base of pyramid
Specifies the first point of the base of the pyramid.

Second point
Specifies the second point of the base of the pyramid.

Third point
Specifies the third point of the base of the pyramid.

Tetrahedron
Choose this option to finish creating the base of the pyramid after specifying the third point to create a three-sided pyramid (tetrahedron).

Set apex of tetrahedron
Specifies the top of the tetrahedron (1).



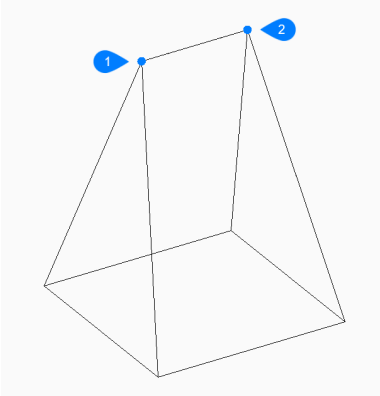
1 Apex

Top surface
Specifies the first point on top surface of tetrahedron, second point and last point of top surface.

Set last point for base
Specifies the fourth point of the base of the pyramid.

Set apex of pyramid
Specifies the location for the top of the pyramid.

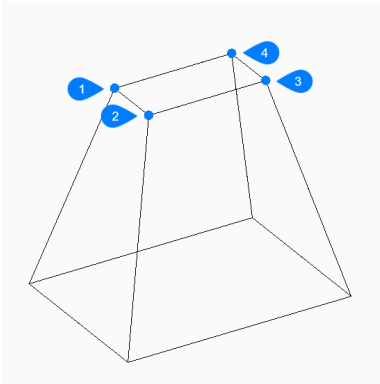
Ridge
Specifies the start and end points to create an edge at the top of the pyramid.



1 Start of ridge

2 End of ridge

Top surface
Specifies four points to create a face at the top of the pyramid.



1 First point

2 Second point

3 Third point

4 Last point

6.34 AI_REVSURF command

This command is obsolete. It still exists for backward compatibility only.

Shape Lite Pro Mechanical BIM

Use REVSURF command instead.

6.35 AI_RULESURF command

This command is obsolete. It still exists for backward compatibility only.

Shape Lite Pro Mechanical BIM

Use the RULESURF command instead.

6.36 AI_SELALL command (Express Tools)

Selects all the entities in the drawing, except for those on frozen layers.

Shape Lite Pro Mechanical BIM

6.37 AI_SPHERE command

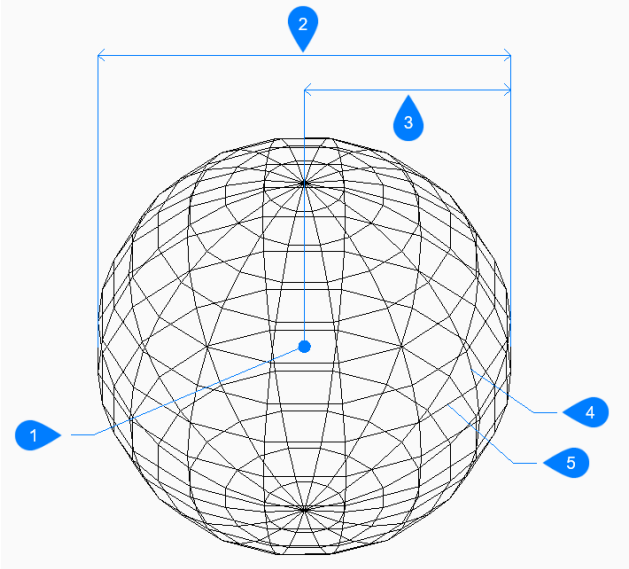
Creates a 3D polygon mesh in the shape of a sphere.

Shape Lite Pro Mechanical BIM

Icon:

6.37.1 Description

Creates a 3D polygon mesh in the shape of a sphere. Choose from a combination of options including center, radius, diameter, and number of segments.



1 Center

2 Diameter

3 Radius

4 Latitudinal segments

5 Longitudinal segments

6.37.2 Method

There is one method to begin creating a sphere:

- Center of sphere

6.37.3 Options within the command

Center of sphere

Specifies the center of the sphere.

Set radius of sphere
Specifies the radius of the sphere.

Diameter
Specifies the diameter of the sphere.

Number of longitudinal segments
Specifies the number of isolines, perpendicular to the xy-plane, used to simulate the sphere's curved surface. The default value is 16.

Number of latitudinal segments
Specifies the number of isolines, parallel to the xy-plane, used to simulate the sphere's curved surface. The default value is 16.

6.38 AI_TABSURF command

This command is obsolete. It still exists for backward compatibility only.

Shape Lite Pro Mechanical BIM

Use the TABSURF command instead.

6.39 AI_TILEMODE1 command (Express Tools)

Sets the TILEMODE system variable to 1.

Shape Lite Pro Mechanical BIM

6.40 AI_TORUS command

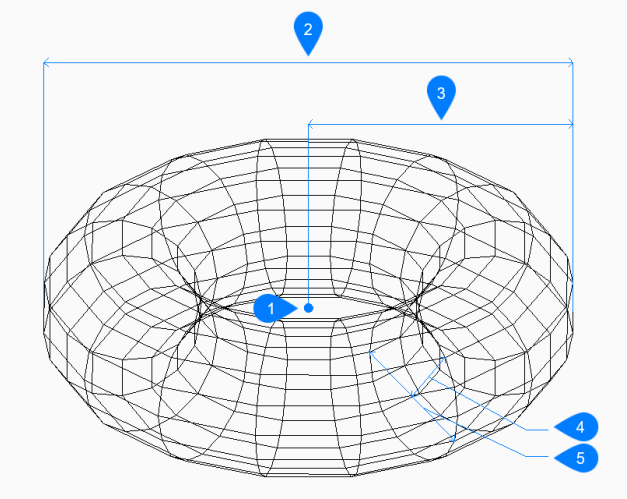
Creates a 3D polygon mesh in the shape of a torus.

Shape Lite Pro Mechanical BIM

Icon: 

6.40.1 Description

Creates a 3D polygon mesh in the shape of a torus. Choose from a combination of options including center and radius or diameter of both the whole torus and the body of the torus.



1 Center

2 Diameter of the whole torus

3 Radius of the whole torus

4 Radius of the body

5 Diameter of the body

6.40.2 Methods

There is one method to begin creating a torus:

- Center of whole torus

6.40.3 Option within the command

Center of whole torus
Specifies the center of the whole torus.

Set Radius of whole torus
Specifies the overall radius of the torus. The radius is measured from the center of the whole torus to the outer edge of the body (tube) of the torus.

Diameter (of whole torus)
Specifies the diameter of the whole torus. The diameter is twice the distance from the center of the whole torus to the outer edge of the body (tube) of the torus.

Set radius of body of torus
Specifies the radius of the body (tube) of the torus.

Diameter (of body of torus)
Specifies the diameter of the body of the torus.

Segments around body of torus
Specify the number of isolines, perpendicular to the xy-plane, used to simulate the torus's curved surface. The default is 16.

Segments around whole torus
Specifies the number of isolines, parallel to the xy-plane, used to simulate the torus's curved surface. The default value is 16.

6.41 AI_WEDGE command

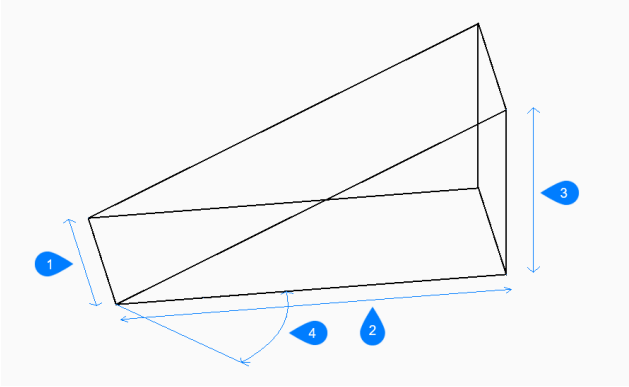
Creates a 3D polyface mesh in the shape of a wedge.

Shape Lite Pro Mechanical BIM

Icon:

6.41.1 Description

Creates a 3D polyface mesh in the shape of a wedge by specifying a corner, the length, width, height and rotation angle.



1 Width

2 Height

3 Length

4 Rotation angle

6.41.2 Method

There is one method to begin creating a wedge:

- Corner of wedge

6.41.3 Options within the command

Corner of wedge
Specifies a corner for the base of the wedge.

Length of wedge
Specifies the length of the wedge.

Width of wedge
Specifies the width of the wedge.

Height of wedge
Specifies the height of the wedge.

Rotation angle for wedge
Specifies the rotation angle of the base of the wedge in the xy-plane.

6.42 AIMLEADEREDITADD command

Adds and removes leaders to mleaders.

Shape Lite Pro Mechanical BIM

Icon: 

6.42.1 Description

Adds and removes one or more leader lines to and from multi-line leaders.

6.42.2 Method

There are two methods to add and remove leaders:

- Add leaders to an existing mleader.
- Remove leaders from an mleader.

6.42.3 Options within the command

Add leaders
New leaders are added to an existing mleader entity.

Remove leaders
One or more leaders are removed from an existing mleader entity.
This command can remove all leaders from an mleader entity, leaving just the text.

6.43 AIMLEADEREDITREMOVE command

Removes and adds leaders to mleaders.

Shape Lite Pro Mechanical BIM

Icon: 

6.43.1 Description

Removes and adds one or more leader lines to and from multi-line leaders.

6.43.2 Method

There are two methods to remove and add leaders:

- Remove leaders from an mleader.
- Add leaders to an existing mleader.

6.43.3 Options within the command

Remove leaders
One or more leaders are removed from an existing mleader entity.
This command can remove all leaders from an mleader entity, leaving just the text.

Add leaders
New leaders are added to an existing mleader entity.

6.44 AIDIMFLIPARROW command

Flips a dimension arrow.

Shape Lite Pro Mechanical BIM

Icon:

6.44.1 Description

Flips the dimension arrows so that they point in the opposite direction.

Even though the prompt refers to plural entities, the command flips one arrow at a time, the one closest to the picked point along the dimension or extension line.

6.45 AIDIMPREC command

Changes the display precision of dimension text.

Shape Lite Pro Mechanical BIM

Icon: 

6.45.1 Description

Increases and decreases the display precision of dimension values retroactively, changing either the number of decimal places or the denominator of fractions. The measured precision remains unchanged.

This command rounds off dimension values up or down, and so it can display false values that seem too high or too low in value. For example, setting AIDIMPREC to 0 on a dimension with a length of 3.525" rounds the display to 4" but the measured length remains 3.525".

6.46 AIDIMSTYLE command

Applies and saves dimension styles.

Shape Lite Pro Mechanical BIM

Icon: 

6.46.1 Description

Saves the current dimension settings as a named style and applies a preset style to selected dimensions in the current drawing.

6.46.2 Method

There are two methods to begin working with dimension styles:

- Apply a saved dimension style to dimensions in the drawing.
- Save dimension settings in the current drawing as a named style.

6.46.3 Options within the command

Apply
Apply an existing dimension style to one or more dimensions.

Note: Use the '-DIMSTYLE command transparently during this command to list the names of dimension styles in the current drawing.

This overrides the changes made to dimensions with commands like AIDIMFLIPARROW and AIDIM-PREC.

Save
Save the style properties of an existing dimension to a named style.
The properties of the saved style can be examined with the DIMSTYLE command.

6.47 ALIGN command

Moves, rotates, and scales entities to align with specified points.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: AL

6.47.1 Description

Moves, rotates and scales entities in 2D or 3D space by specifying a set of alignment points.

6.47.2 Options within the command

Select entities
Selects entities that are to be aligned.

Note: Do not select destination entities, otherwise they are also moved.

Specify first source point
Specifies the source alignment point.

Specify first destination point
Specifies the destination alignment point.

Note: If you press Enter at this prompt, the source object moves.

Note: Specify a second pair of source and destination points to move and rotate entities.

Specify second source point
Specifies the second source alignment point.

Specify second destination point
Specifies the matching destination point.

Note: Press Enter to move and rotate 2D entities.

Note: Specify a third pair of source and destination points to move and rotate 3D entities.

Scale entities based on alignment points
Allows you to scale the source entity.

Note: Scaling is only available when aligning entities using two pairs of points.

Yes
Scales objects relative to the picked points.

No
Does not scale objects.

Specify third source point
Specifies the source point for 3D alignment.

Specify third destination point
Specifies the matching destination point.

6.48 ALIASEDIT command (Express Tools)

Creates, edits and removes BricsCAD command aliases.

Shape Lite Pro Mechanical BIM

6.48.1 Method

Opens the **Customize** dialog box, **Command Aliases** tab to create, edit and remove command aliases.

6.49 ALIGNMENT command

Creates Horizontal and 3D alignments.

Shape Lite Pro Mechanical BIM

Icon: 

6.49.1 Description

Creates Horizontal and 3D alignments to represent a reference system used to position linear objects in space.

Note: You can offset Horizontal Alignments. The result of the OFFSET command on Horizontal Alignments is a non-associative polyline.

Horizontal Alignment is created by specifying points of intersection (PI) of the tangent polygon. Curves are drawn automatically between the tangents.

Note: The CVDEFAULTCURVETYPEHA system variable specifies the curve type to be used when create a new horizontal alignment or adding new PI. The default value is **Auto Spiral-Curve-Spiral**.

6.49.2 Options within the command

Select TIN surface
Allows you to select a TIN surface used to drape Horizontal alignment and to create 3D alignment.

Note: Two 3D alignments are created: one represents the projection of the Horizontal Alignment to the TIN Surface and second represents the Vertical Alignment, which is calculated based on the **Vertical tolerance** parameter.

Pick pi point
Allows you to pick the PI points for your Horizontal alignment.

Note: If **Select TIN surface** option was previously selected, the 3D alignment is automatically generated.

Undo
Undoes the last PI point.

Specify start station
Allows you to define the start station when start creating a new alignment.

6.50 ALIGNMENT3D command

Creates a **3D Alignment** from a specified vertical alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.50.1 Description

Creates a **3D Alignment** entity by selecting a vertical alignment.

6.51 ALIGNMENTCURVE command

Creates unconstrained curve element on existing or on a new **Horizontal Alignment**.

Shape Lite Pro Mechanical BIM

Icon: 

6.51.1 Method

There are two methods:

- Create a new curve element on existing horizontal alignment.
- Create a new alignment by drawing a curve element.

The unconstrained curve element can be created on **existing** or on a **new Horizontal Alignment** in two ways. In both ways an existing horizontal alignment needs to be selected first. The user can then either specify a curve centre point in the drawing or select **Points** option from the command line. If the user has specified a centre point, he determines the orientation of the curve element in the next step, and at the end specify its diameter. Otherwise, if option **Points** has been selected in the command line, the user specifies

three points that define the new curve element. User can then continue drawing new curve elements on existing horizontal alignment until presses the Enter key.

6.51.2 Options within the command

Select existing horizontal alignment
Creates a new curve element on existing alignment.

Create new alignment
Creates new curve element on existing **Horizontal Alignment**..

Specify curve direction
Toggles between clockwise and counterclockwise direction.

Points
Creates new curve element of the **Horizontal Alignment** by specifying its start point, second point and end point.

6.52 ALIGNMENTEDIT command

Edits an alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.52.1 Method

Select a Horizontal or Vertical alignment to edit.

6.52.2 Options within the command for Horizontal alignment

Continue with Pi
Allows you to continue the Horizontal alignment.

Note: This option works only if the last element is fixed line.

Delete Element
Allows you to delete alignment elements.

Add Pi
Adds a Pi Point to the alignment.

Remove Pi
Removes a Pi point.

Change Tin surface
Changes the Tin surface of the Horizontal alignment by adding additional TIN surface.

Add Station equation
Allows you to add station equation.

Remove Station equation
Allows you to remove station equation by specifying the index.

6.52.3 Options within the command for Vertical alignment

Add Pvi
Adds a Pvi point to the alignment.

Remove Pvi
Removes a PVI point.

Change Tin surface
Changes the Tin surface of the Vertical alignment by adding additional TIN surface.

6.53 ALIGNMENTELEMENTBETWEEN command

Creates a new element between two selected alignment elements.

Shape Lite Pro Mechanical BIM

Icon: 

6.53.1 Description

The new element is drawn tangential to both attachment elements.

Note: Therefore, we say that such an element is constrained by both selected elements.

Depending on the selected type of the new element, some parameters of the element are maintained when attachment elements are edited.

The following types of new elements can be created between two existing elements: lines, curves, spirals, spiral-curve-spiral combinations, spiral-spiral combinations and the spiral-line-spiral combinations.

6.53.2 Options within the command

Line
Creates new line element between two existing curve elements. The new line is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.
The line can be added between the following element combinations:

- Between two existing unconstrained circles.
- Between two existing constrained circles as a part of combinations, such as a Curve with Spiral and a Curve-Spiral-Spiral combination.
- Between an existing unconstrained and constrained circle.

Note: The way the line is drawn between the selected elements depends on their directions.

Curve
Creates a new curve element between two existing alignment elements. The new curve is drawn tangential to both attachment elements. The tangency and the curve radius are maintained when attachment elements are edited.
The position of points on (attachment) elements, between which new curve is created and the curve length are adjusted according to the editing of attachment elements.
The curve can be added between all element combinations except between the first selected element and the constrained spiral.

Note: The curve is defined by the type of attachment elements, specified curve radius and solution angle (< or >180°). The way the circle is drawn between the selected elements depends on the attachment elements direction.

Spiral

Creates new spiral element between two existing alignment elements. The new spiral is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.

Note: A free spiral is defined by the type and the direction of selected attachment elements. Since the spiral has only one geometric solution, the user cannot specify parameters, such as parameter A or the spiral length.

SCS spiral-curve-spiral

Creates new spiral-curve-spiral combination between two existing alignment elements. The new combination is drawn tangential to both attachment elements. The tangency, curve radius and both spiral lengths are maintained, when attachment elements are edited.

The position of points on (attachment) elements, between which new combination is created and the curve length are adjusted according to the editing of attachment elements.

Select attachment elements, specify curve radius and spiral lengths to add spiral-curve-spiral combination between selected alignment elements.

SS spiral-spiral

Creates new spiral-spiral combination between two existing curve elements with the same orientation and possibly different radii. The new combination is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.

Spiral parameters (length and parameter A) and the position of points on (attachment) elements, between which new combination is created, are calculated according to the specified A1/A2 (or L1/L2) ratio.

Note: The position of points on (attachment) elements, between which new combination is created and the curve length are adjusted according to the editing of attachment elements.

SLS spiral-line-spiral

Creates new spiral-line-spiral combination, with a specified spiral lengths, between two existing curve elements. The new combination is drawn tangential to both attachment elements.

Note: The position of points on (attachment) elements, between which new combination is created, and spiral parameters are adjusted according to the editing of attachment elements.

Specify first entity

Allows you to select the first alignment element.

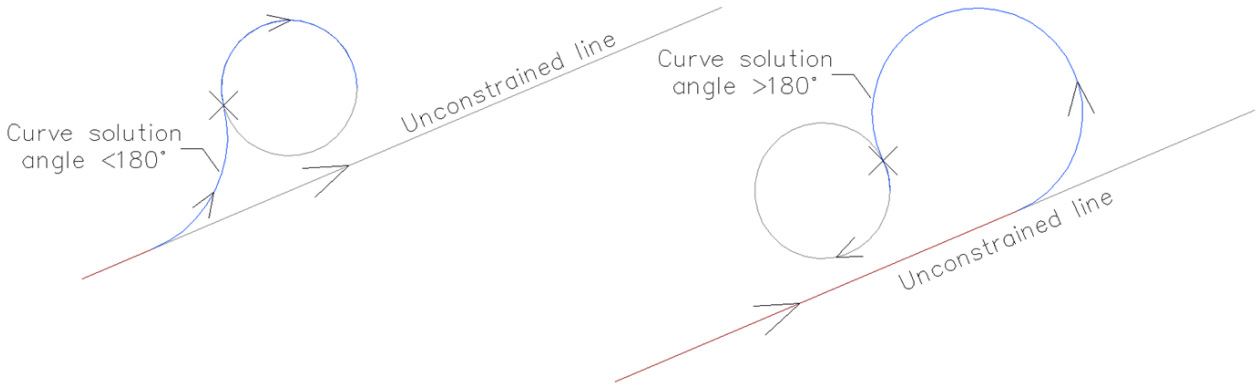
Specify second entity

Allows you to select the second alignment element.

Is curve solution angle

Toggles between **Greaterthan180** and **Lessthan180**.

The figure below shows two possible solutions for a curve with an angle < 180° (left) and > 180° (right) added between an unconstrained line and an arc element.

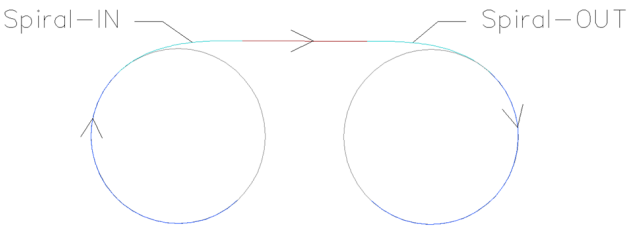


spiral parameter A
Allows you to specify the A1/A2 ratio.

Spiral length
Allows you to specify the L1/L2 ratio.

Spiral in-length
Allows you to specify the spiral-in length.

Spiral out-length
Allows you to specify the spiral-out length.
The figure below shows the spiral-line-spiral combination, created between two unconstrained curves, where the user specifies the spiral-IN and spiral-OUT lengths.



Specify radius
Allows you to specify the radius of the new spiral and curve element.

6.54 ALIGNMENTELEMENTFROM command

Attaches new alignment element before (to) or after (from) the existing alignment element.

Shape Lite Pro Mechanical BIM

Icon:

6.54.1 Description

The following types of new elements can be attached to existing elements: lines, curves, spirals, lines with spirals, curves with spirals and curve-spiral-spiral combination.

Note: New element is always drawn tangential to the specified point on the selected attachment element.

6.54.2 Options within the command

Line
Attach a new line element before (to) or after (from) the selected alignment element.

Note: Line length can be either specified in the drawing or its value can be entered in the command line.

Curve
Attach a new curve element before (to) or after (from) the selected alignment element.
To attach a curve before the existing element, select it closer to the element start point. Otherwise click closer to attachment element end point to attach curve after (starting from) the element.

Note: The curve geometry is defined by the curve radius, curve angle solution (greater or less than 180°) and the pass-through point.

Spiral
Attach a new spiral element before (to) or after (from) the selected alignment element.
To attach a spiral before the existing element, select it closer to the element start point. Otherwise click closer to the attachment element end point, to attach curve after (starting from) the element.

Note: The spiral geometry is defined by the radius, length and the direction (clockwise or counter-clockwise).

LS line with spiral
Attach a combination before (to) or after (from) the selected alignment element. As the name implies, a combination consists of two elements, a spiral and a line.

Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select attachment element closer to its start point. In this way we get the **Line-Spiral** combination. Otherwise, if existing element is selected closer to its end point, then we get the **Spiral-Line** combination after the attachment element. Such combination starts by spiral transition to a line, which passes through a specified pass-through point (line end point).

CS curve with spiral
Attach a **Curve-Spiral** combination before (to) or after (from) the selected alignment element. As the name implies, a combination consists of two elements, a spiral and a curve.

Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select existing element closer to its start point. In this way we get the **Curve-Spiral** combination. Otherwise, if attachment point is selected closer to element end point, then we get the **Spiral-Curve** combination. Such combination starts by spiral transition to a curve, which passes through a specified pass-through point.

First select the attachment element, then specify spiral radius and length, then specify the curve solution angle (greater or less than 180°) and last specify the curve pass-through point to draw the **Curve-Spiral** combination.

CSS curve-spiral-spiral
Attach a **Curve-Spiral-Spiral** combination before (to) or after (from) the selected alignment element.
As the name implies, a combination consists of two oppositely directed spirals and a curve that passes through a specified point.

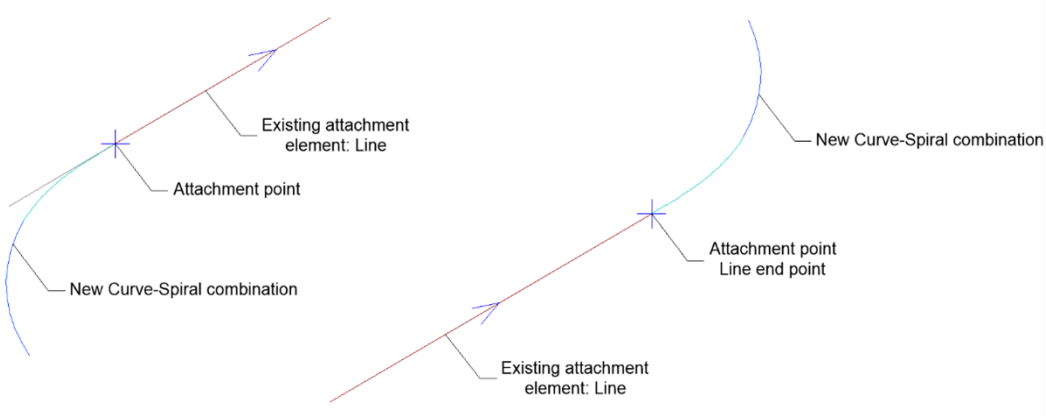
Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select attachment element

closer to its start point. In this way we get the **Curve-Spiral-Spiral** combination. Otherwise, if existing element is selected closer to its end point, then we get the **Spiral-Spiral-Curve** combination.

First select the attachment element, then specify spiral-in length and radius, then specify spiral-out length and specify curve pass-through point in the last step to draw **Curve-Spiral-Spiral** combination.

Element

Select this option to attach new element (or combination of elements) to the specified point on the attachment element. Select attachment element closer to its start point to attach new element before (to) the attachment element. Or select attachment element closer to its end point to attach new element after (from) attachment element.



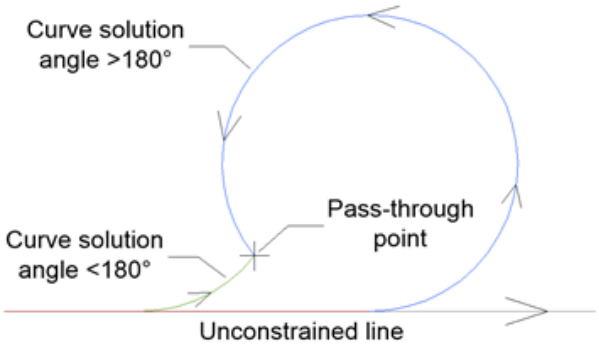
element END

Select this option to attach new element (or combination of elements) to the start/end point of the attachment element. Select attachment element closer to its start point to attach new element before (to) the attachment element. Or select attachment element closer to its end point to attach new element after (from) attachment element.

Is curve solution angle

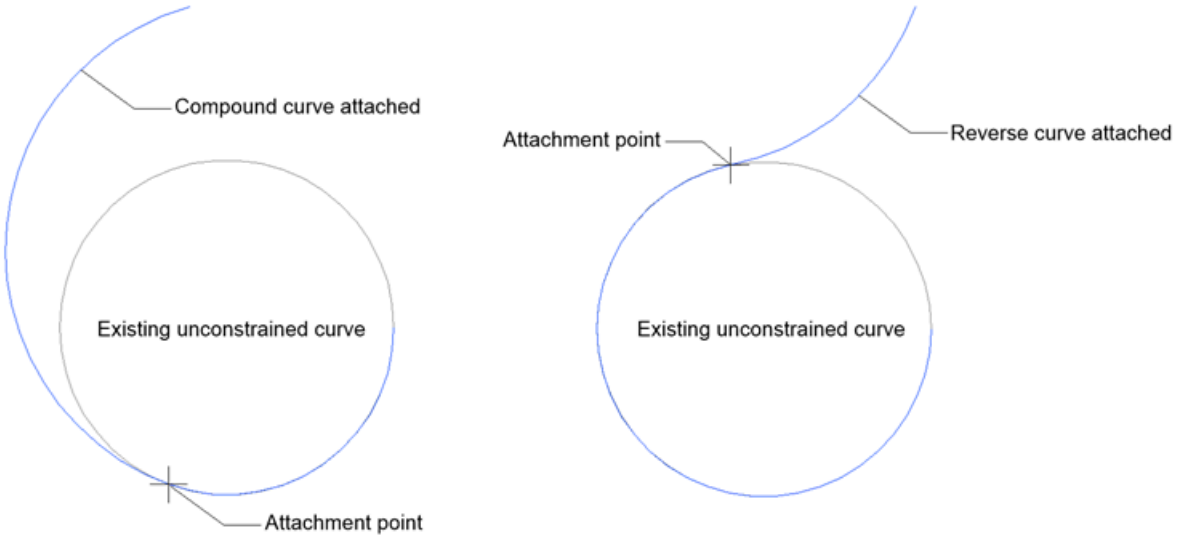
Toggles between **Greaterthan180** and **Lessthan180**.

The figure below shows two possible solutions for the curve passes through the same specified point, depends on the selected option for the curve solution angle.



Is curve compound or reverse?

Toggles between **Compound** and **Reverse** curve attached to existing unconstrained curve element, as illustrated in the figure below:



Length
Allows you to specify the new line length.

Specify spiral length
Allows you to specify the new spiral length.

Specify radius
Allows you to specify the radius of the new spiral and curve element.

Specify curve direction
Specifies the new element orientation. Toggles between clockwise and counter-clockwise orientation.

6.55 ALIGNMENTLINE command

Creates unconstrained line element between two specified points on existing or on a new **Horizontal Alignment**.

Shape Lite Pro Mechanical BIM

Icon:

6.55.1 Method

There are two options:

- Create a new line element between two specified points on existing horizontal alignment.
- Create a new alignment by drawing a line element between two specified points.

6.55.2 Options within the command

Select existing horizontal alignment
The new unconstrained line element is created on selected existing Horizontal Alignment, between two specified points.

Note: First select the existing horizontal alignment in the drawing, and then specify the first and second point, between which a new line element is created. You can then continue drawing new line elements on existing horizontal alignment until you press the Enter key.

Create new alignment
Creates new **Horizontal Alignment** by drawing a line element between two specified points.

Note: Specify the first and second point in the drawing, between which a new line element is created. You can continue drawing new line elements until press the Enter key.

6.56 ALIGNMENTVIEW command

Creates a vertical alignment view for the selected horizontal alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.56.1 Method

Select a horizontal alignment, previously created with ALIGNMENT command, and a point as the origin for vertical alignment view.

6.57 ALIGNMENTVERTICAL command

Creates **Vertical Alignment**.

Shape Lite Pro Mechanical BIM

Icon: 

6.57.1 Description

Creates **Vertical Alignment** on a selected **Vertical Alignment View**. It also automatically draws the **3D Alignment** of the corresponding alignment.

6.57.2 Method

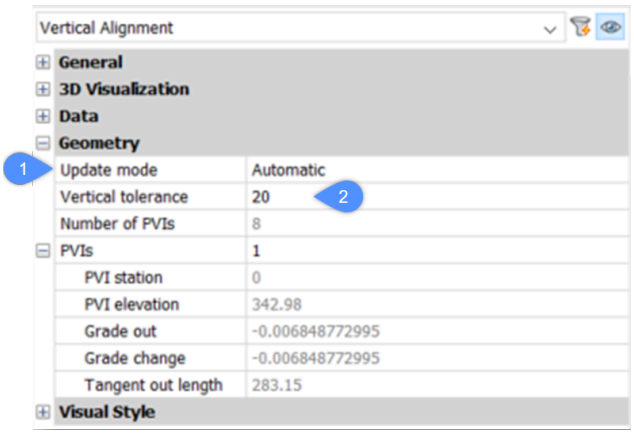
There are two methods:

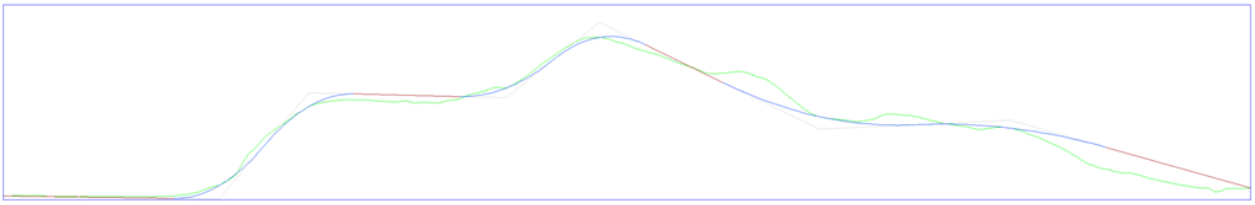
- Creates **Vertical Alignment** by picking PVI points.
- Creates **Vertical Alignment** automatically.

6.57.3 Options within the command

Undo
Undoes the last specified vertical alignment PVI point.

Automatic 3d alignment
Automatically creates **Vertical Alignment** and **3D Alignment**.





The Automatic Vertical Alignment is calculated so that the elevation difference to the terrain line does not exceed the value of the **Vertical tolerance** parameter, which is set to 0.1 by default.

Once the **Vertical Alignment** is created, you can edit the value of the **Vertical tolerance** (2) parameter in the **Properties** panel.

Additionally, the value of the **Update mode** (1) parameter is set to **Automatic**. This means that you cannot edit the geometry of the Automatic Vertical Alignment. To edit it, set the **Update mode** parameter to **Manual**.

6.58 ALIGNSPACE command

Adjusts viewport angle, zoom factor, and pan position.

Shape Lite Pro Mechanical BIM

Note: This command can be entered transparently during commands ('alignspace).

Note: This command works only in paper space.

Note: The model space viewports must have PERSPECTIVE mode OFF.

6.58.1 Description

Adjusts viewport angle, zoom factor, and pan position by selecting points in the same model space viewport and paper space.

6.59 AMBALLOON command

Adds balloon annotations for geometry, blocks or standard parts annotated with part references.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.59.1 Method

Note:

- The balloon annotation is added to the AM_5 layer.
- The balloon annotations are compatible with the legacy AutoCAD® Mechanical application.

Select the part references from the current BOM for which you want to add balloons.

6.59.2 Options within command

auTo
Adds balloon annotations for all selected part references at once.

Note: The starting points for the balloon arrowhead leaders are inside the part reference symbols.

Horizontal
Places the balloons horizontally.

Vertical
Places the balloons vertically.

Angle
Places the balloons in a specified direction.

One
Adds balloon annotation for one-by-one selected part reference.

Note: You can select the starting point for the balloon arrowhead leader in the drawing.

set Bom
Allows you to choose which BOM to use.

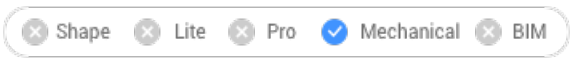
Note:

- When no Bill of Materials (BOM) is present in the drawing, a BOM named **MAIN** will be automatically generated.
- Part references that belong to the chosen BOM are highlighted in red to be easily selected.
- The part reference balloon number is according to the BOM used.

?
Lists the existing BOM in the drawing on the Command line.

6.60 AMBOMSETTINGS command

Launches the **BOM Settings** dialog box.



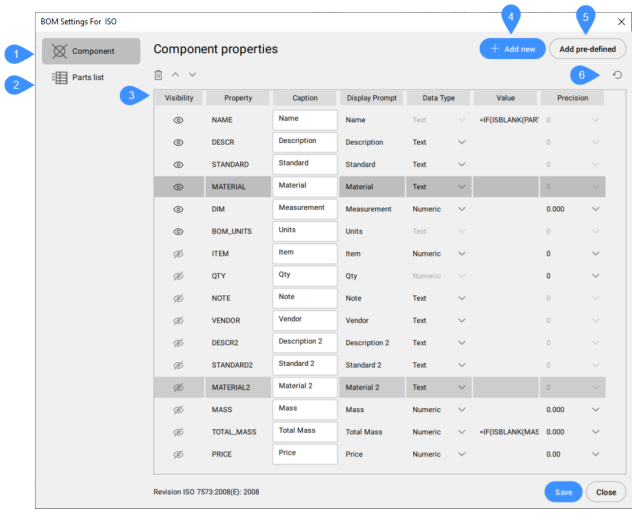
Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.60.1 Method

Opens the **BOM Settings**dialog box.

Configures part references (Part References) properties and set up default properties for parts lists, and data capture for the BOM.



1 Component

2 Parts list

3 List of properties

4 Add new

5 Add pre-defined

6 Restore defaults

6.60.2 Component

Contains a list of component properties that are visible in Part Reference. More properties can be selected by pressing the Ctrl / Shift key.

Above this list there are delete, move up and move down icons, which are available only when a row is selected or the selected row(s) can be moved up or down.

6.60.3 Parts list

Contains a list of **Parts list** properties that are visible in the Parts Lists.

6.60.4 List of properties

Visibility

Toggles the visibility of the property in the **Part Reference** and **Parts List** dialog boxes.

Property
Displays the default name of a component property. This unique name is the one by which a BOM column is referred to in formulas.

Caption
Defines the column header displayed in the Parts List, overriding the default property name. It is editable and will appear as the component property name in the **Part Reference** and **Parts List** dialog boxes.

Note: The changes are saved in the drawing. To use them later, save the drawing as a template (DWT) file and use this template when you start a new drawing.

Display prompt
Displays the chosen column header to be displayed for this property in a BOM table or parts list.

Data type
Defines the data type for the property (Numeric or Text)

Value
Defines the default value for the property. You can also use a formula to define the property.

Precision
Defines the precision for the property.

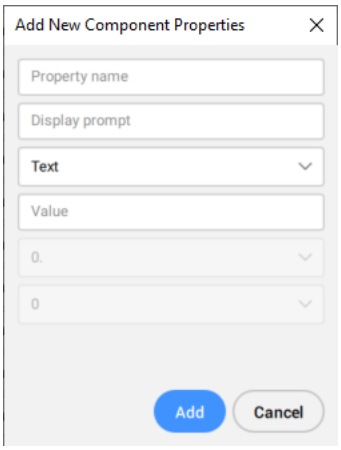
Caption Align
Defines the column header position in the cell.

Value Align
Defines the value position in the cell.

Width
Defines the width of the cell.

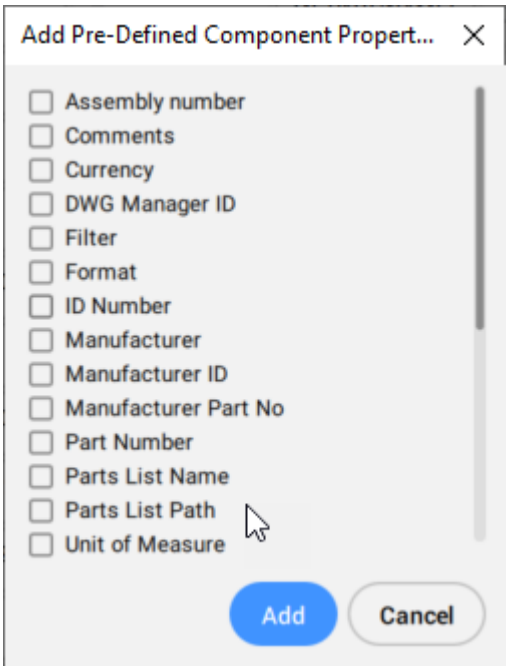
6.60.5 Add new

Adds new component properties as columns to BOM. Opens **Add New Component Properties** dialog box.



6.60.6 Add pre-defined

Adds new pre-defined component properties columns to BOM. Opens **Add Pre-Defined Component Properties** dialog box.



6.60.7 Restore defaults

Resets all settings to the default value for the current drafting standard.

6.61 AMCLEAN command

Saves a mechanical 2D drawing as a non-mechanical drawing.

Shape Lite Pro Mechanical BIM

6.61.1 Description

Cleans the drawing from Mechanical 2D data to be able to be exported to a DXF file.

6.61.2 Method

This command is available when the drawing contains mechanical 2D entities and the LOADMECHANICAL2D system variable is ON. The command saves the drawing that contains mechanical 2D entities to a non-mechanical drawing so that it can be exported to DXF file format.

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

Opens the **Save file as non-mechanical** dialog box to choose a file name for the non-mechanical drawing.

6.62 AMDATUMID command

Draws a datum identifier and attaches it to an entity.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.62.1 Method

Select the object to attach the datum identifier symbol and specify the points for its location, then the **Datum Identifier** dialog box opens. The first leader segment is perpendicular to the attached object.

When you double-click a datum identifier symbol, the **Datum Identifier** dialog box opens, allowing you to change its parameters.

After creating the datum identifier symbol with the AMDATUMID command, their properties can be changed also in the **Properties** panel:

Symbol	
Drafting standard	DIN
Standard revision	DIN EN ISO 1101:2008, DIN EN ISO 5459:
Scale	1
Requirements	
Datum Area	Near side
Identifier	1
Thread note	A
Datum note	B
Leader	
Primary arrowhead	DatumBlank45
Surface arrowhead	
Offset from Object	0
Extension beyond leader	0

Symbol

Drafting standard
Displays the drafting standard.

Standard revision
Displays the standard revision.

Scale
Sets the note scale.

6.63 AMEDGESYM command

Draws an edge symbol and attaches it to an entity.

Shape Lite Pro Mechanical BIM

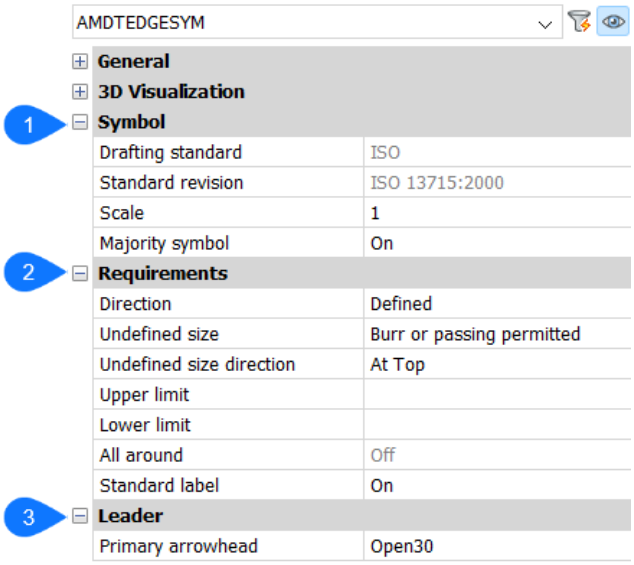
Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.63.1 Method

Select the object to attach the edge symbol to and specify the points for its location.

After creating the edge symbol with the AMEDGESYM command, its properties can be configured in the **Properties** panel:



Symbol

Drafting standard
Displays the drafting standard.

Standard revision
Displays the standard revision.

Scale
Sets the note scale.

Majority symbol
Chooses to display or to not display the majority symbol.

Requirements
Defines the edge symbol requirements.

Direction
Defines the position of the **Undefined size** sign.

Defined
The sign will be on a position defined by the **Undefined size direction** property.

Undefined
The sign will be on an undefined (middle) position.

Undefined size
Defines the edge.

None
Defines no requirements for the edge type. If available, upper and lower limits are displayed.

Burr or passing permitted
Defines the edge type as burr or passing.

Undercut required
Defines the edge type as undercut.

Undefined size direction
Defines the position of the undefined size sign.

Upper limit
Defines the upper value by typing it in the box or by selecting it from the drop-down list.

Lower limit
Defines the lower value by typing it in the box or by selecting it from the drop-down list.

All around
Adds the all around mark to the edge symbol.

Standard label
Displays the standard revision next to the symbol.

Leader

Primary arrowhead
Sets the leader style.

6.64 AMFCFRAME command

Creates a feature control frame and attaches it to an entity.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.64.1 Description

Allows you to create a feature control frame symbol that can be attached to an entity in the drawing. A feature control frame is rectangular and divided into at least two sections.



6.64.2 Method

Select the entity to attach the feature control frame symbol and specify the points for its location. By pressing Enter, the **Feature Control Frame** dialog box opens.

Double-click an existing feature control frame symbol to open the **Feature Control Frame** dialog box to change its properties. Also, the properties of the feature control frame symbol can be configured in the **Properties** panel.

6.65 AMNOTE command

Draws a leader note and attaches it to an entity.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.65.1 Method

Select the object to attach the leader note, specify the points for its location and write the text note. The text of the note should be formatting in the **Text Formatting** dialog box that opens.

After creating it, you can change the note text in the **Contents** field of the **Properties** panel.

After creating the leading note with the AMNOTE command, their properties can be configured in the **Properties** panel:

Symbol	
Drafting standard	DIN
Standard revision	DIN 6780:2000-10
Scale	1
Flip symbol	Right aligned
Show reference line	Off
Text justification	Align Left
Leader justification	To Reference Line
Requirements	
Contents	
Leader	
Primary arrowhead	By Standard

Symbol

Drafting standard
Displays the drafting standard.

Standard revision
Displays the standard revision.

Scale
Sets the note scale.

Flip Symbol
Switches between Right and Left aligned text from the ending of the extension line.

Show reference line
Switches between show (ON) and hidden (OFF) the reference line.

Text justification
Specifies the location of the text regarding the extension line.

Leader justification
Sets the leader justification type.

Requirements

Content
Allows you to edit the content of the note.

Leader

Primary arrowhead
Sets the leader style.

6.66 AMPARTLIST command

Inserts in the drawing area a parts list, according to an indicated BOM.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.66.1 Method

Note: The Part List is added to the AM_5 layer.

Specify according to which BOM you will create the parts list and place it in the drawing area. The **Parts List** dialog box opens to configure appearance and content of the table.

Note: Multiple Parts Lists from the same BOM containing different information are allowed.

6.66.2 Options within the command

Main
Selects the main BOM for the part list.

Note: The **Main** BOM table is current by default. The last used BOM table in the command is set as current.

?
Lists all available BOM in the command panel.

6.67 AMPARTREF command

Creates a Part Reference symbol and places it in the drawing area.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.67.1 Method

The command is available only for drawings that contain mechanical entities.

Note: The Part Reference symbols is added to the AM_12 layer.

Select the object to attach a Part Reference symbol to it.

The **Part reference** dialog box opens.

Note:

- If you click an empty space in the drawing area, the program places an unattached Part Reference symbol (a Part Reference that is not attached to any geometry) at that position. The **Part reference** dialog box opens.
- In the **Properties** panel, you can edit the Part Reference symbol properties.
- When you double-click the part reference symbol, the **Part reference** dialog box opens.

6.67.2 Options within the command

Block
Applies a Part Reference to a Block Reference. The **Part reference** dialog box opens.

Copy
Copies a selected Part References to another place in the drawing area.

Reference
Duplicates the selected part reference and keeps them linked.

6.68 AMPARTREFEDIT command

Edits Part Reference properties.

Shape Lite Pro Mechanical BIM

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.68.1 Method

Note: The Part Reference symbols is added to the AM_12 layer.

Select the Part Reference symbol, and then the **Part reference** dialog box displays.

Note: The Part References symbols from the entire drawing are highlighted.

6.69 AMPOWERDIM_ALI command

Creates aligned linear dimensions as ACM entities.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.69.1 Description

Creates aligned linear dimensions. This command is a shortcut to the **Linear Aligned** option of the PDIM command.

6.69.2 Method

Define the two extension lines or select the entity and specify the location of the dimension. The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

Note:

- When SNAP is ON, the dimension line freezes during placement to the snap distance and is marked with red color.
- Press **Enter** to be able to select an entity.
- Press **Esc** to finish the command.

6.69.3 Options within the command

Placement options
Specifies the object to include for distance snap calculation.

Options
Opens the **Placement Options** dialog box to set the **Snap value**.

6.70 AMPOWERDIM_ANG command

Creates angular dimensions as ACM entities.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.70.1 Description

This command is a shortcut to the **Angular** option of the PDIM command.

6.70.2 Method

There are three methods to define the angle:

- By defining the three points.
- By defining the two lines.
- By defining an arc or a circle.

The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

After the angular dimension is created, you can define new dimensions by choosing PDIM command options.

6.70.3 Options within the command

Angle
Sets the angle of the dimension text.

Text
Overrides the dimension text.

6.71 AMPOWERDIM_DIA command

Creates a diameter dimension as ACM entities.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.71.1 Description

Creates diameter dimensions for arcs and circles. The command is a shortcut to the **Radial Diameter** option of the PDIM command. It automatically adds a diameter symbol to the dimension text.

6.71.2 Method

Select the arc or the circle and the location of the dimension line.

The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

6.71.3 Options within the command

Angle
Sets the angle of the dimension text.

Text
Overrides the dimension text.

6.72 AMPOWERDIM_HOR command

Creates linear horizontal dimensions as ACM entities.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.72.1 Description

The command is a shortcut to the **Linear Horizontal** option of the PDIM command.

6.72.2 Method

Define the two extension lines or select the entity and specify the location of the dimension. The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

Note:

- When SNAP is ON, the dimension line freezes during placement to the snap distance and is marked with red color.
- Press **Enter** to be able to select an entity.
- Press **Esc** to finish the command.

6.72.3 Options within the command

Placement options
Specifies the object to include for distance snap calculation.

Options
Opens the **Placement Options** dialog box to set the snap value.

6.73 AMPOWERDIM_RAD command

Creates radial dimensions for arcs and circles as ACM entities.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.73.1 Description

This command is a shortcut to the **Radial Radius** option of the PDIM command. It automatically adds a radius symbol to the dimension text.

6.73.2 Method

Select the arc or the circle and the location of the dimension line.

The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

6.73.3 Options within the command

Angle
Sets the angle of the dimension text.

Text
Overrides the dimension text.

6.74 AMPOWERDIM_ROT command

Creates rotated linear dimensions as ACM entities.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.74.1 Description

It measures the distance between two points in a direction that is at an angle to a line connecting the two points. This command is a shortcut to the **Linear Rotated** option of the PDIM command.

6.74.2 Method

Define the two extension lines or select the entity and specify the location of the dimension. The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

Note:

- When SNAP is ON, the dimension line freezes during placement to the snap distance and is marked with red color.
- Press **Enter** to be able to select an entity.
- Press **Esc** to finish the command.

6.74.3 Options within the command

Placement options
Specifies the object to include for distance snap calculation.

Options
Opens the **Placement Options** dialog box to set the snap value.

6.75 AMPOWERDIM_VER command

Creates vertical linear dimensions.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.75.1 Description

The command is a shortcut to the **Linear Vertical** option of the PDIM command.

6.75.2 Method

Define the two extension lines or select the entity and specify the location of the dimension. The **Edit Dimensioning** dialog box opens to define the parameters of the dimension.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

Note:

- When SNAP is ON, the dimension line freezes during placement to the snap distance and is marked with red color.
- Press **Enter** to be able to select an entity.
- Press **Esc** to finish the command.

6.75.3 Options within the command

Placement options
Specifies the object to include for distance snap calculation.

Options
Opens the **Placement Options** dialog box to set the snap value.

6.76 AMPOWEREDIT command

Identifies the selected dimension or mechanical symbol entity and executes the most relevant command to edit it.

Shape Lite Pro Mechanical BIM

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.76.1 Description

The AMPOWEREDIT command identifies any dimension or mechanical symbol entity you select and executes the most appropriate command to edit it. For example, if you run the command and select a dimension entity, the **Edit Dimensioning** dialog box opens.

Note: The AMPOWEREDIT command is also launched when you double-click one of the dimension or mechanical symbol entities.

6.77 AMRESCALE command

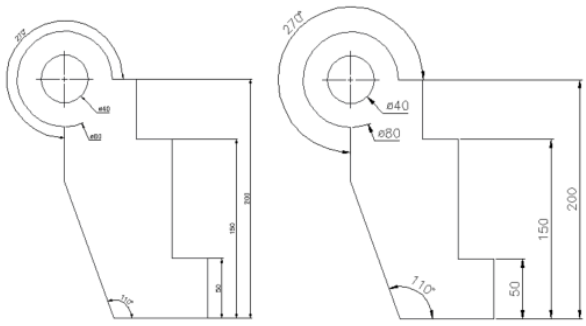
Rescales dimensions, symbols, tables and texts in both model space and layout.

Shape Lite Pro Mechanical BIM

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.77.1 Method

Enter the new symbol scale and select the entities you want to rescale. Entities rescaled in both model space and layout are dimensions, symbols, hatches, and tables. Annotations are enlarged or shrunk without affecting the drawing geometry. Also, the offset distance between the dimension lines and the entity is adjusted to prevent scaled dimension texts from overlapping. The mechanical symbol associated with the PDIM entity (for example, the datum identifier) remains attached and moves with the PDIM entity, keeping the same side.

Before AMRESCALE	After AMRESCALE factor scale 2
	

6.78 AMSETUPDWG command

Changes the current standard of a mechanical 2D drawing or set up a mechanical 2D template for a non-mechanical drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.78.1 Method

After opening a drawing which contains mechanical entities there are two use cases:

1 The standard of the current mechanical drawing can be changed by selecting another DWT mechanical2D template file from the **Choose one of the Mechanical 2D template file** dialog box.

2 The standard of a non-mechanical drawing can be changed with a standard of a mechanical drawing, by selecting a DWT Mechanical2D template file from the **Choose one of the Mechanical 2D template file** dialog box. The mechanical-related data, like dictionaries, style tables, etc. will be initialized.

Note:

- A message with the name of a new standard will be displayed on the Command line.
- If select a template without a defined mechanical standard, an error message is displayed in the Command line: **You did not choose a mechanical 2D template**.
- If the mcad2d functionality is not available, an error message will be displayed in the Command line.

6.79 AMSIMPLEWELD command

Creates caterpillars and side weld annotations.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.79.1 Options within the command

Front

Creates a front view fillet or single -V butt welds (caterpillars) along a curve (line, polyline, arc, ellipse) between two specified points.

Note: Caterpillars are created as block references in the layout.

Alignment

Defines the type of the caterpillars weld.

Left
Adds a partial caterpillar and places it on the left of the curve.

Center
Adds a full caterpillar and centers it on the curve.

Right
Adds a partial caterpillar and places it on the right of the curve.

Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the **Ctrl** key to cycle the above various options.

Width
Defines the weld width.

Note:

- For a partial caterpillar, the weld width is the distance between the endpoint of the beam and the selected edge.
- For a full caterpillar, the weld width is the distance between the two endpoints of the beam.

Step
Defines the bead pitch.

switch Orientation
Toggles the bead orientation.

switch Direction
Toggles the bead direction.

Full
Creates the front weld symbol along the entire selected curve.

Side
Creates a side view seam or single -V butt welds with the specified Leg width and rotation angle.

Fillet
Creates a side view for a fillet weld.

single - V butt
Creates a side view for a single - V butt weld.

Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the **Ctrl** key to cycle through the above options.

change leg Width
Defines another value for the leg width.

6.80 AMSURFSYM command

Creates a surface symbol.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.80.1 Method

Note: The surface symbol is added to the AM_5 layer.

Select the object to attach the surface symbol and specify the points for its location. The **Surface texture** dialog box opens, to define all surface texture parameters.

When you double-click a surface texture symbol, the **Surface texture** dialog box opens, allowing you to change its parameters.

You can also change the surface texture symbol's properties in the **Properties** panel:

Symbol	
Drafting standard	DIN
Standard revision	DIN EN ISO 1302:2002
Scale	1
Symbol type	Material removal required
Majority symbol	On
Requirements	
All around	Off
Machine allowance	
Process note	Process Note1
Process note 2	Process Note 2
First requirement	Ra 6,3
Second requirement	Ra 6,3
Third requirement	9
Direction of lay	Crossed in two oblique directions
Leader	
Offset from Object	0
Extension beyond leader	0

Note: The surface symbol properties depend on the used standard.

6.81 AMWELDSYM command

Creates a welding symbol in the drawing space.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.81.1 Method

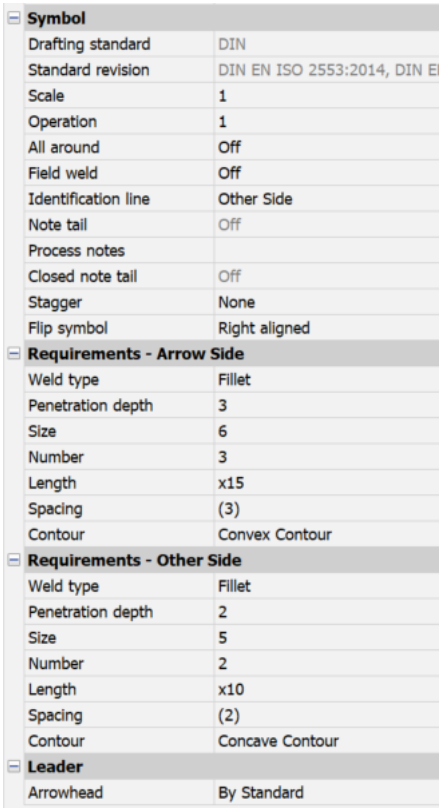
Note: The welding symbol is added to the AM_5 layer.

Select the entity to attach the welding symbol and specify the points for its location. The **Weld Symbol** dialog box opens, where you can customize the weld symbol parameters.

Note: To place welding symbols in the drawing views (Paper space), use BMWELDSYMRETRIEVE command.

When you double-click a welding symbol, the **Weld Symbol** dialog box opens, allowing you to change its parameters.

Also, you can change the welding symbol's properties from the **Properties** panel:



Symbol	
Drafting standard	DIN
Standard revision	DIN EN ISO 2553:2014, DIN EI
Scale	1
Operation	1
All around	Off
Field weld	Off
Identification line	Other Side
Note tail	Off
Process notes	
Closed note tail	Off
Stagger	None
Flip symbol	Right aligned
Requirements - Arrow Side	
Weld type	Fillet
Penetration depth	3
Size	6
Number	3
Length	x15
Spacing	(3)
Contour	Convex Contour
Requirements - Other Side	
Weld type	Fillet
Penetration depth	2
Size	5
Number	2
Length	x10
Spacing	(2)
Contour	Concave Contour
Leader	
Arrowhead	By Standard

6.82 AMTAPERSYM command

Creates a slope symbol in the drawing space.

Shape Lite Pro Mechanical BIM

Icon:

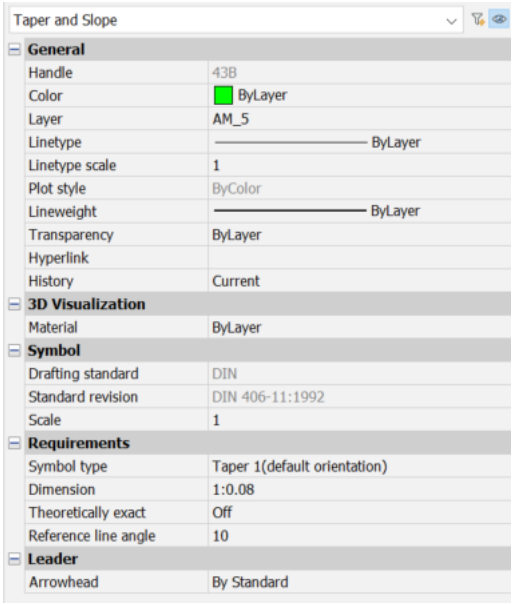
Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

6.82.1 Method

Note: The slope symbol is added to the AM_5 layer.

Select a taper centerline or a slope baseline, then select a taper or slope object to attach and specify the start and next points for the slope symbol location. The gradient is automatically calculated.

The **Properties** panel contains information about the drawing standard and the properties of the slope symbol.



6.83 ANALYTICS command

Opens the **BricsCAD Analytics Program** dialog box.

Shape Lite Pro Mechanical BIM

6.83.1 Description

Opens the **BricsCAD Analytics Program** dialog box which allows you to improve BricsCAD by sending pseudonymised diagnostics and usage data to Bricsys.

6.84 ANIMATIONEDITORCLOSE command

Closes the Animation editor panel.

Shape Lite Pro Mechanical BIM

6.84.1 Description

Closes the **Animation editor** panel to hide it from the current workspace. If the **Animation editor** panel is stacked when you close it, the **Animation editor** tab or icon is removed from the stack.

6.85 ANIMATIONEDITOROPEN command

Opens the **Animation Editor** panel.

Shape Lite Pro Mechanical BIM

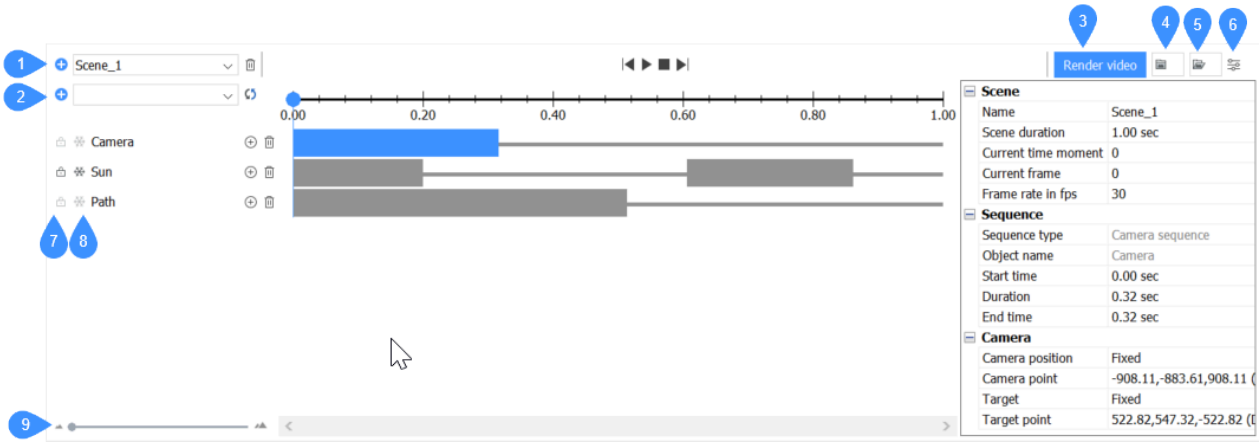
Icon:

6.85.1 Description

Opens the **Animation Editor** panel to display it in the current workspace. The **Animation Editor** panel appears at the same size and on the same location as it did before it was closed or collapsed. Like any other dockable panel, the **Animation Editor** panel can be either floating, docked or stacked.

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

Note: The **Animation Editor** panel is disabled on macOS and Linux.



1 Add scene

2 Add timeline

3 Render video

4 Record graphics for the current scene

5 Load animation graphics

6 Configurator

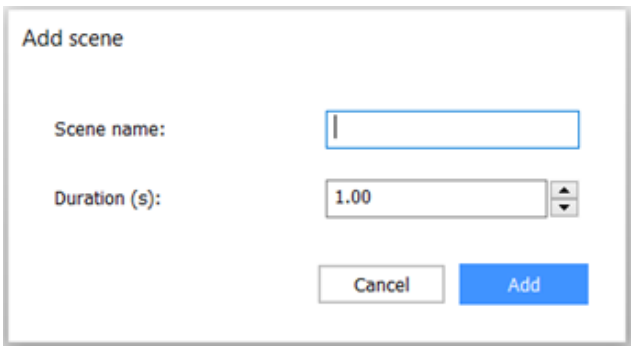
7 Lock/unlock timeline button

8 Disable/enable timeline effects button

9 Slider

6.85.2 Add scene

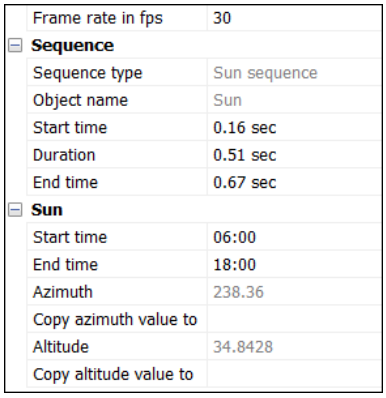
Opens the **Add scene** dialog box to specify the scene name and its duration (s):



6.85.3 Add timeline

Adds the selected parameter to the timeline.

One of the parameters that can be added is the sun effect. Once the sun animation sequence is added to the timeline, several properties are available in the properties configurator of the **Animation Editor** panel.



Sequence properties
Defines the properties of the sun sequences on the timeline.

Sun properties
Defines the properties of the sun itself regarding its position and timing.

6.85.4 Render video

Opens the **Render video** dialog box to specify the render options.

6.85.5 Record graphics for the current scene

Records animation graphics and saves it to a zip-archive or to a folder of your choice.

6.85.6 Load animation graphics

Loads animation graphics data from a zip archive or a folder and opens the special mode for playing these graphics.

6.85.7 Configurator

Toggles the **settings** panel.

6.85.8 Lock/unlock timeline button

Toggles the timeline to lock any timeline changes or vice versa, in order to enable changes.

6.85.9 Disable/enable timeline effects button

Toggles the timeline effects between enabled and disabled. Disabled timeline effects are excluded from the animation.

6.85.10 Slider

Allows you to move the slider to zoom in/out on the timelines.

6.86 ANIMATIONEDITORPLAYBACK command

Loads animation graphics.

Shape Lite Pro Mechanical BIM

6.86.1 Description

Loads animation graphics data from a zip-archive or a folder.

Note: This command is only executed if the **Animation Editor** panel is active. After the graphics are loaded, **Animation Editor** is switched to a special animation graphics playback mode.

6.87 -ANIMATIONEDITORPLAYBACK command

Loads animation graphics via Command line.

6.87.1 Description

Loads animation graphics data from a zip-archive or a folder, with the **Animation Editor** panel closed. Playback options for graphics are available in the Command line prompt until the command is closed.

Note:

- Adding and modifying camera effects in animation graphics playback mode is not supported for this command.
- When **Animation Editor** panel is active, the command has the same functionality as ANIMATIONEDITORPLAYBACK command.

6.87.2 Method

Choose a Zip-file or Folder to load animation graphics to your drawing.

6.87.3 Options within the command

Play
Starts the playback for the animation.

paUse
Pauses the playback.

Stop
Stops the playback.

Render
Opens the **Render video** dialog box that allows you to set the render options for your animation.

Cancel
Cancels the command.

6.88 ANIMATIONEDITORRECORD command

Records animation graphics.

Shape Lite Pro Mechanical BIM

6.88.1 Description

Records animation graphics and saves it to a zip-archive or to a folder of your choice. After selecting the needed save option, the recording process will start. Depending on the complexity of the model, this can take a long time.

6.88.2 Options within the command

Zip-file
Opens the **Select a zip-file name to save animation graphics** dialog box that allows you to save the animation graphics to a zip-archive.

Folder
Opens the **Select a folder to save animation graphics** dialog box that allows you to save the animation graphics to a folder of your choice.

6.89 ANIPATH command

Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

Shape Lite Pro Mechanical BIM

Icon: 

6.89.1 Description

Displays the **Motion Path Animation** dialog box.

6.90 ANNORESET command

Resets moved annotative scale representations.

Shape Lite Pro Mechanical BIM

Icon: 

6.90.1 Description

The locations of scale representations linked with annotative entities can be moved with grips. This command returns selected scale representations to their default positions.

6.91 ANNOUPDATE command

Updates the selected annotative entities to match their current annotative style.

Shape Lite Pro Mechanical BIM

Icon: 

6.91.1 Method

Select one or more annotative entities, such as text, dimensions, hatches, blocks to update them to their current annotative styles. If all the entities in the drawing are selected, the program will ignore non-annotative entities.

6.92 APPARENT command

Toggles the **Apparent intersection** entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PLANVIEWINT

6.92.1 Description

Toggles the **Apparent intersection** entity snap to enable or disable snapping to apparent intersection. You can launch this command at the Command line to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

6.93 APPLOAD command

Opens the Load application files dialog box.

Shape Lite Pro Mechanical BIM

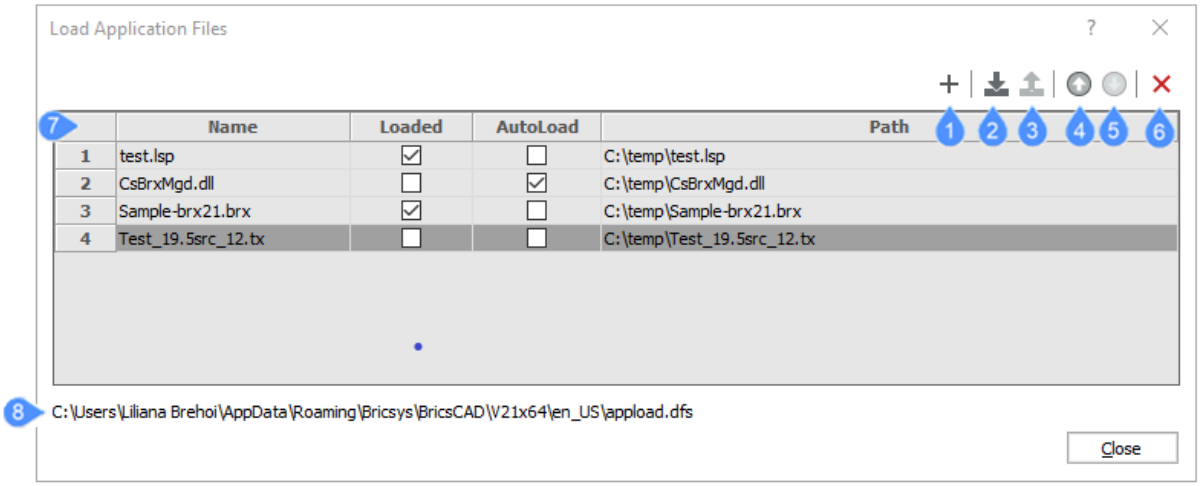
Icon: 

6.93.1 Description

Opens the **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.
- DVB - 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

6.93.2 Add application file

Opens the **Select Application Files** dialog box.

6.93.3 Load selected application

Loads the selected application(s) into BricsCAD. Only compatible applications with BricsCAD will be loaded.

6.93.4 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.

6.93.5 Move application upward

Moves the selected application(s) up in the list.

6.93.6 Move application downward

Moves the selected application(s) down in the list.

6.93.7 Remove selected application

Removes the selected application(s) from the list.

6.93.8 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.

6.93.9 Application File Path

Displays the location folder for the appload.dfs file that contains all applications added, using the **Select Application Files** dialog box. You can open the file using Notepad.

Applications listed in appload.dfs are loaded on start-up.

6.94 ARC command

Creates an arc.

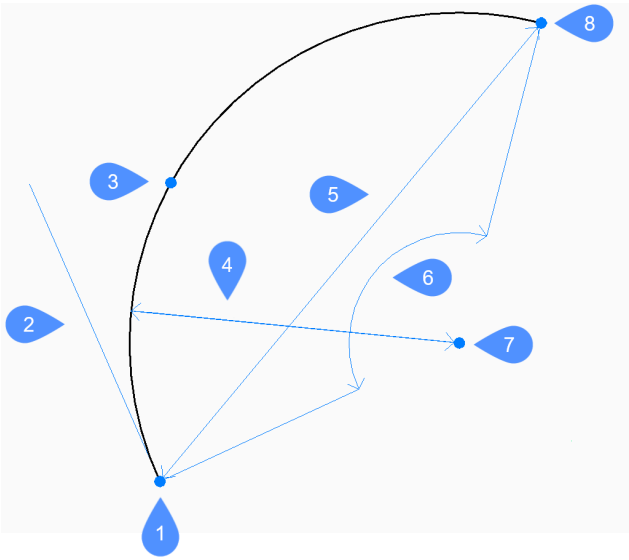
Shape Lite Pro Mechanical BIM

Icon: 

Alias: A

6.94.1 Description

Creates an arc from a combination of options including the start point, second point, end point, center, included angle, direction, and length of chord.



1 Start

2 Direction

3 Second

4 Radius

5 Length of chord

6 Angle

7 Center

8 End

6.94.2 Method

This command has 3 methods to begin creating an arc:

- Start of arc
- Center
- Follow last point

You can press the **Ctrl** key while drawing an arc to change its direction from counterclockwise to clockwise.

Start of arc
Allows you to create an arc using three specified points. The first point represents the start point.

Second point
Specifies the second point along the arc's circumference.

End point
Specifies the end point of the arc.

Angle
Allows you to create a counterclockwise arc from the start point, using a center point with a specified included angle.

Center
Creates a counterclockwise arc from the starting point to an endpoint that lies on an imaginary ray drawn from the center point through the second point specified.

Length of chord
Creates either a minor or a major arc based on the length of a straight line between the start point and endpoint.

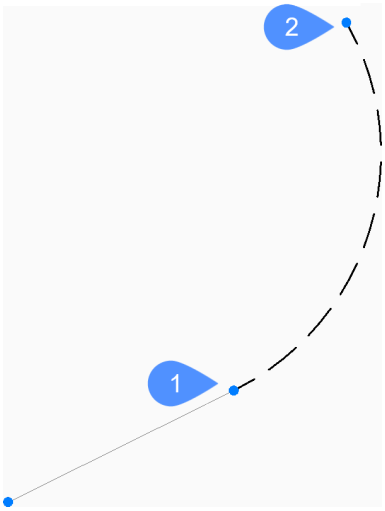
Direction
Creates minor or major arcs, clockwise or counterclockwise, based on initial direction for the arc.

End
Creates an arc based on a specified endpoint.

Radius
Creates a minor counterclockwise arc from the start point to the end point.

Follow last point
Creates an arc from the last drawn arc or line segment, following its angle.

End of arc
Specifies the end point of the arc. The arc is drawn tangent to the previous segment.



1 Last point

2 End of arc

6.95 ARCTEXT command (Express Tools)

Adds text aligned with a selected arc.

Shape Lite Pro Mechanical BIM

Icon:

6.95.1 Description

Adds text aligned with a selected arc or edits the text of an existing arc aligned text entity.

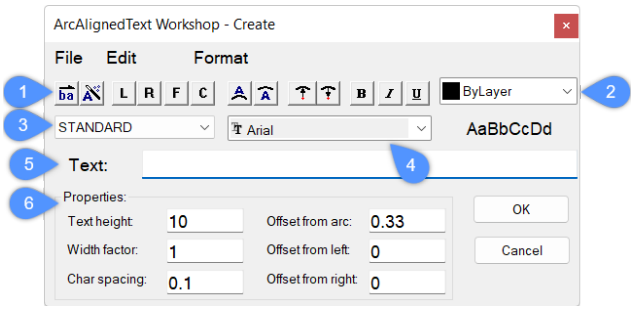
6.95.2 Method

Select an arc or an arc aligned text entity.

Note: You cannot use this tool with splines, polylines or circles.

The **ArcAligned Text Workshop - Create** dialog box opens, which allows you to customize the display of the text.

The **ArcAlignedText Workshop - Create** dialog box allows you to design arc aligned text entities.



1 Text options

2 Text color

3 Text style

4 Text font

5 Text content

6 Properties

6.95.3 Text options

- : reverses text reading order.
- : controls the text behavior when the arc is moved.
- : aligns the text to the left.
- : align the text to the right.
- : fits the text along the arc.
- : centers the text along the arc.
- : places the text on the convex side.
- : places the text on the concave side.
- : places the text outwards from center.
- : places the text inwards to center.
- : makes the text bold.
- : makes the text italic.

- : makes the text underlined.

6.95.4 Text color

Sets the color of the text.

6.95.5 Text style

Specifies the text style.

6.95.6 Text font

Specifies the text font.

6.95.7 Text content

Allows you to type the text to be displayed.

6.95.8 Properties

Text height
Specifies the text height.

Width factor
Specifies the width of the text in proportion to the length of the arc.

Char spacing
Specifies the space between characters.

Offset from arc
Specifies the offset distance from the arc to the text.

Offset from left
Specifies the offset distance from the left end of the arc.

Offset from right
Specifies the offset distance from the right end of the arc.

6.96 AREA command

Finds the area and perimeter of 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AA

6.96.1 Method

Specify the points that define the area for which BricsCAD will report the area and perimeter.

Note: You can stop the calculation by pressing ESC.

6.96.2 Options within the command

find area of one Entity
Provides the area and/or perimeter of the selected entity.

Note: The information reported by this command varies according to the entity selected:

- Line and sketch - length
- Arc and elliptical arc – area and length
- Circle and ellipse - area and circumference/perimeter
- Closed polyline and closed spline - area and perimeter
- Open polyline and spline - area and length
- 3D objects - surface area

Add areas together
Allows to add up the areas of two or more entities.

Subtract areas
Allows to remove areas of entities from the total area and perimeter.

Note: This option can only be used after you have found the area of at least one entity.

6.97 ARRANGE command

Spatially organizes a set of entities by aligning and/or distributing them along the axes of the current UCS.

Shape Lite Pro Mechanical BIM

Icon: 

6.97.1 Methods

The ARRANGE command spatially organizes a set of entities (2D and/or 3D) by aligning and/or distributing them along the axes of the current UCS. You can perform as many arrange operation as you want on the original set of entities with a single run of the command.

This command creates a new arrangement of the selected entities. The arrangement of the entities is defined based on the bounding boxes of the selected entities (i.e., the smallest box which contains the entity), with respect to the selection bounding box (i.e., the smallest box which contains all selected entities).

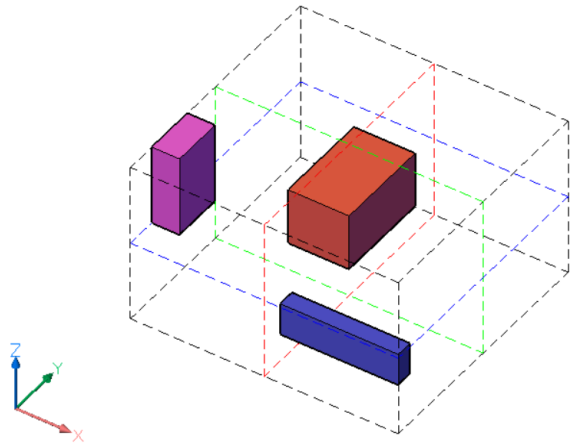
To make a new arrangement with the selected entities, the ARRANGE command uses the following elements:

- A selection of entities (these can be both 2D and 3D).
- A direction for the arrangement. This can be a main axis (X, Y, or Z) of the current UCS.
- The bounding box of the selection.
- The bounding boxes of each entity in the selection.
- The arrangement option selected by the user.

The arrangement of the selected entities is done with the following approach:

To create a new configuration of the selected entities, the ARRANGE command uses some bounds. First, there are the main bounds, that are attached to the selection bounding box. Then, there are the secondary bounds, that are attached to the bounding boxes of each entity in the selection.

This is a simple example of a selection bounding box and three other bounding boxes for the entities in the selection:



The selection bounding box is represented by black dashed lines. The bounds of the selection bounding box are defined by its six faces, two for each axis of the current UCS. There are also three center bounds, one for each axis, located in the center of the selection bounding box. The center bounds are represented by colored dashed lines, red for X axis, green for Y axis and blue for Z axis.

The colored boxes are the bounding boxes of the selected entities. For each entity in the selection, there are six outside bounds, defined by the faces, and three center bounds. The bounds of an entity are defined similarly to the ones of the selection bounding box.

In summary:

- The selection of entities has a total of nine bounds. There are three bounds for each of the axes of the UCS. The notation of these bounds are **1** (Center), **2** (Lower), and **3** (Upper), for each of the axes.
- Also, each entity has nine bounds, similar to the ones of the selection.

For each axis, the bounds are defined as follows:

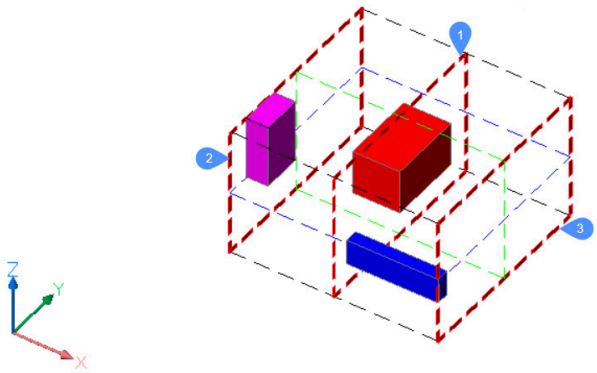
2 - the Lower bound is defined by the face of the bounding box placed at the smallest coordinate along the specified axis.

3 - the Upper bound is defined by the face of the bounding box placed at the largest coordinate along the specified axis.

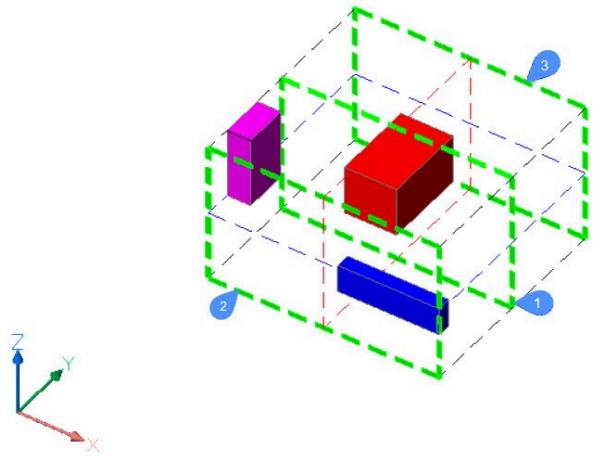
1 - the Center bound is located between the **2** (Lower) and **3** (Upper) bounds at the same distance from each of them.

In the pictures below, the bounds of the selection bounding box are highlighted with thick dashed lines.

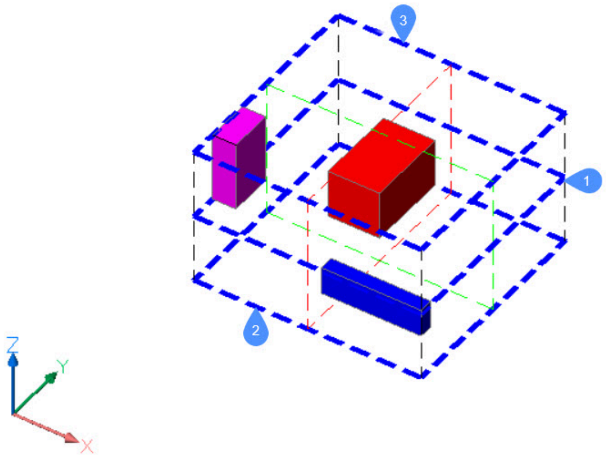
For the X axis:



For the Y axis:



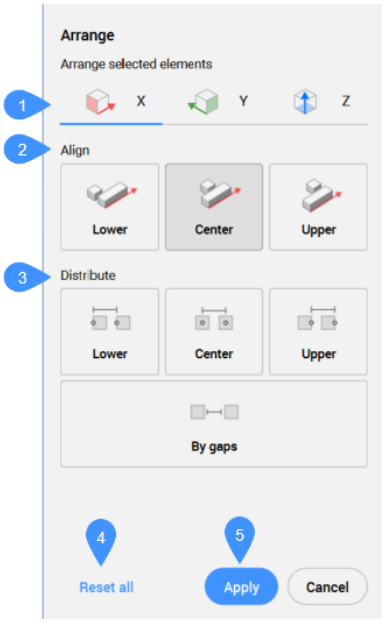
For the Z axis:



To define the new position of an entity in the selection, the ARRANGE command uses the bounds of the selection, the bounds of the entity, and the method selected by the user.

6.97.2 Arrange command context panel

The **Arrange** command context panel opens after finishing selecting entities.



1 Direction

2 Align

3 Distribute

4 Reset all

5 Apply

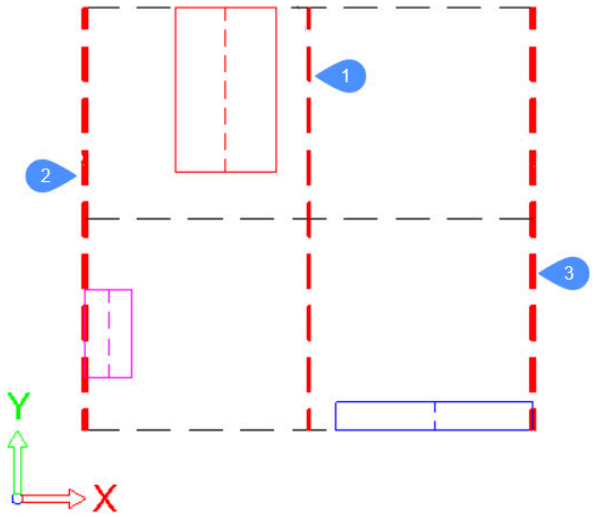
6 ***Note:*** The options within the command are the same options as in the **Arrange** command context panel.

Note: The Command line can still be used to arrange the selected entities while the **Arrange** command context panel is active.

6.97.3 Direction

Defines the direction along which the entities are arranged. The possible directions are the three main axes of the UCS.

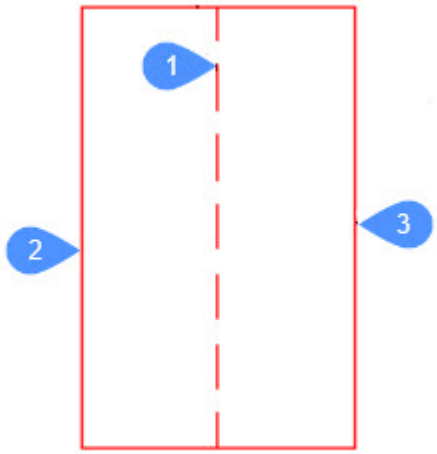
X-direction
As example, this is a simplified 2D view, in XY plane, of the situation described above:



The main bounds, attached to the selection bounding box, are represented by thick dashed red lines.

For each entity in the selection, its bounding box is represented by continuous lines, each with a different color. In this case, for each entity, the bounds, **2 (Lower)**, **1 (Center)** and **3 (Upper)**, will be the left side, center line and right side of its bounding rectangle, respectively.

For the red entity, the bounds will be like this:



Y-direction

The bounds are defined in a similar way as in the X-direction.

Z-direction

The bounds are defined in a similar way as in the X-direction.

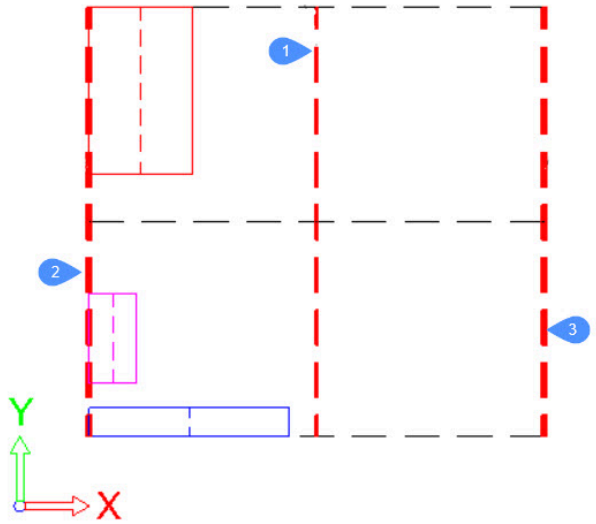
6.97.4 Align

None

The entities will not be arranged along the specified direction. If an arrangement was already set along the specified direction during the execution of the command, the entity will be moved to its original position on the specified axis.

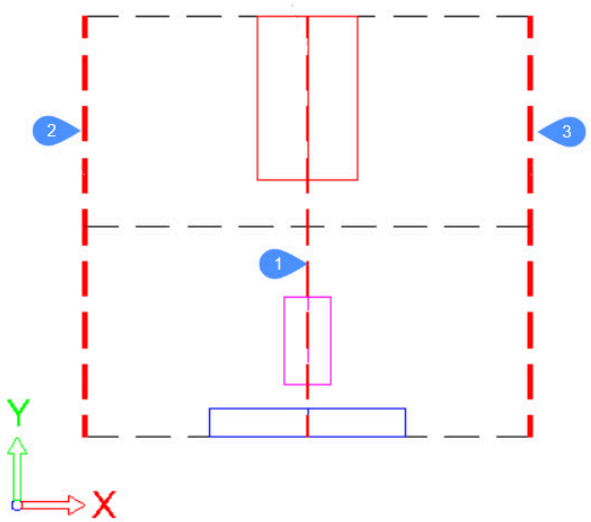
Lower Align

The lower bound of each entity will be aligned with the lower bound of the selection bounding box in the specified direction.



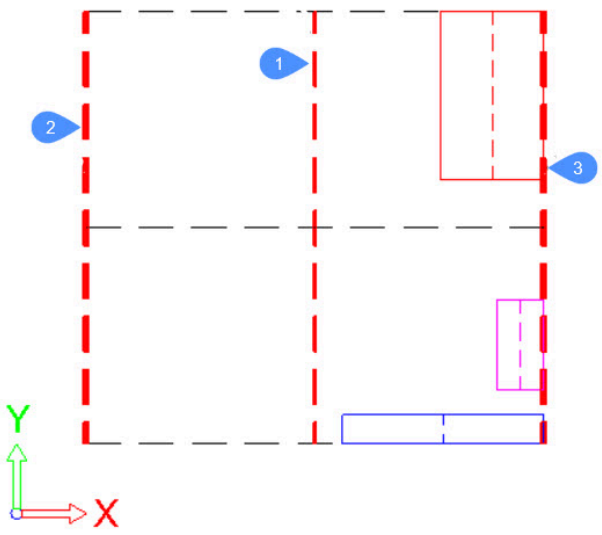
Center Align

The center bound of each entity will be aligned with the center bound of the selection bounding box in the specified direction.



Upper Align

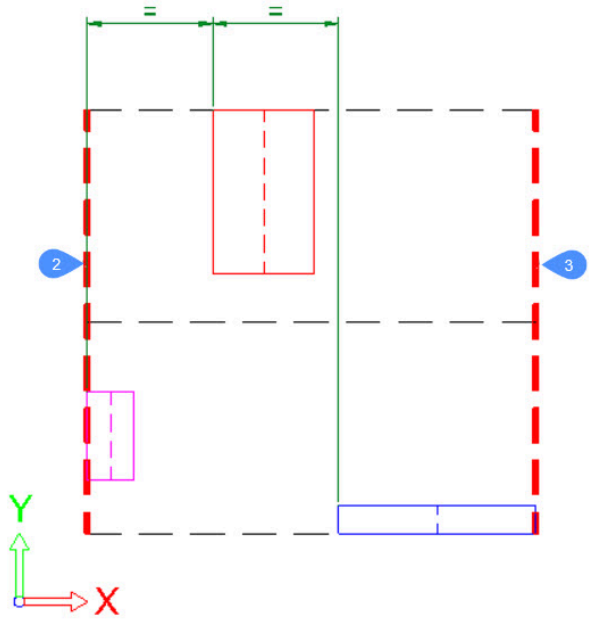
The lower bound of each entity will be aligned with the lower bound of the selection bounding box in the specified direction.



6.97.5 Distribute

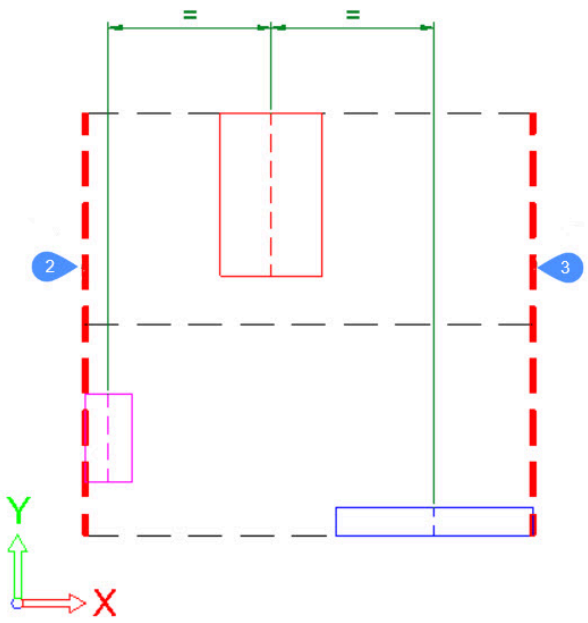
Lower (Equally spaced)

The distance between the lower bounds of consecutive entities will be the same along the specified direction.



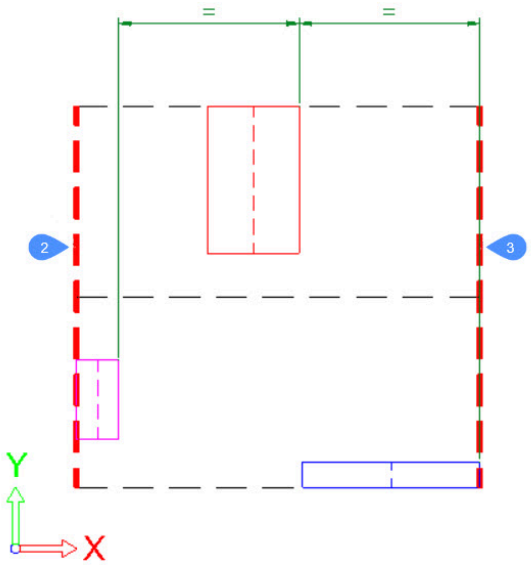
Center (Equally spaced)

The distance between the center bounds of consecutive entities will be the same along the specified direction.



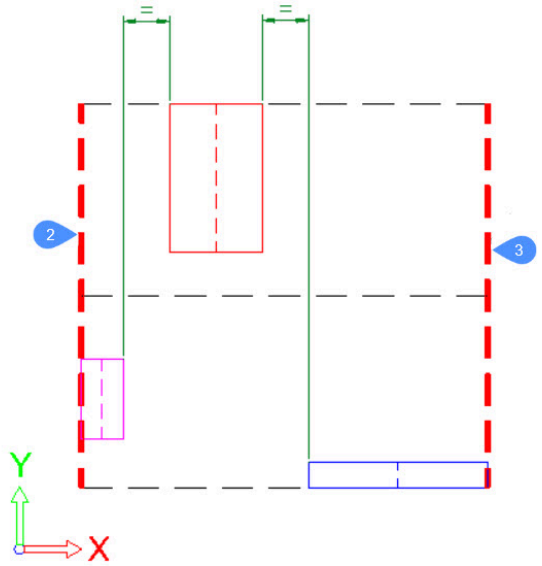
Upper (Equally spaced)

The distance between the lower bounds of consecutive entities will be the same along the specified direction.



By gaps/Equally spaced

The gaps between the entities will have the same size along the specified direction.



6.97.6 Reset all

Resets all entities to their initial position as before the command was launched.

6.97.7 Apply

Applies the rearrangement and closes the command.

6.98 ARRAY command

Creates an array of entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AR

6.98.1 Description

Creates a polar or rectangular array, or along a path of entities through the command line.

Note: Arrays can be created using 2D or 3D entities.

6.98.2 Methods

There are three methods to create an array of entities:

- Rectangular
- Path
- Polar

Note: The default array type is saved by the ARRAYTYPE variable.

6.98.3 Options within the command

Rectangular
Distributes entity copies into any number of rows, columns and levels (on Z direction).

PAth
Distributes entity copies evenly along a path into multiple rows and levels.

POlar
Distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.

6.99 -ARRAY command

Creates an array of entities.

Shape Lite Pro Mechanical BIM

6.99.1 Description

Creates a static polar or rectangular array of entities through the Command line.

Note: Arrays can be created using 2D or 3D entities.

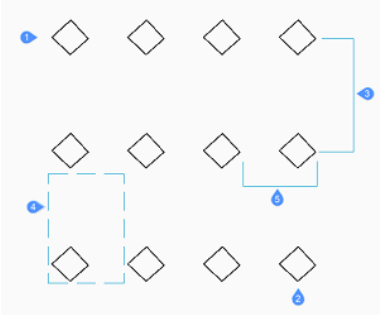
6.99.2 Method

There are two methods to create an array of entities:

- Polar
- Rectangular

6.99.3 Options within the command

Rectangular
Creates linear, rectangular, or square arrays.



Number of rows in the array
Specifies the number of rows. (1)

Note: Enter 1 for a linear array and a negative number to draw the array downwards.

Number of columns
Specifies the number of columns. (2)

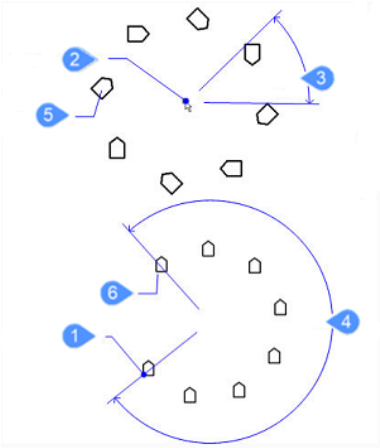
Note: Enter a number value to draw the array to the left.

Vertical distance between rows
Specifies the distance between rows of entities. (3)

Spacing rectangle
Specifies two points that indicate the spacing between rows and columns. (4)

Horizontal distance between columns
Specifies the distance between columns of entities. (5)

Polar
Creates polar (circular) arrays.



Base
Relocates the base point of the selected entities. (1)

Center of polar array
Specifies the center point. (2)

Number of items to array
Specifies the number of items in the array.

Note: Enter a number larger than 1.

enter to specify Angle between items
Specifies the angle between each copy. (3)

Angle to fill
Determines the extent and direction of the array. (4)

Note: Enter 360 for a full circle of copies or a smaller number to draw a partial polar array. Enter a positive angle to draw the array counterclockwise or a negative number to draw the array clockwise.

Rotate entities around the array?
Determines whether copies are rotated when they are arrayed.

Yes
Rotates entities around the array. (5)

No
Does not rotate entities around the array. (6)

6.100 ARRAYCLASSIC command

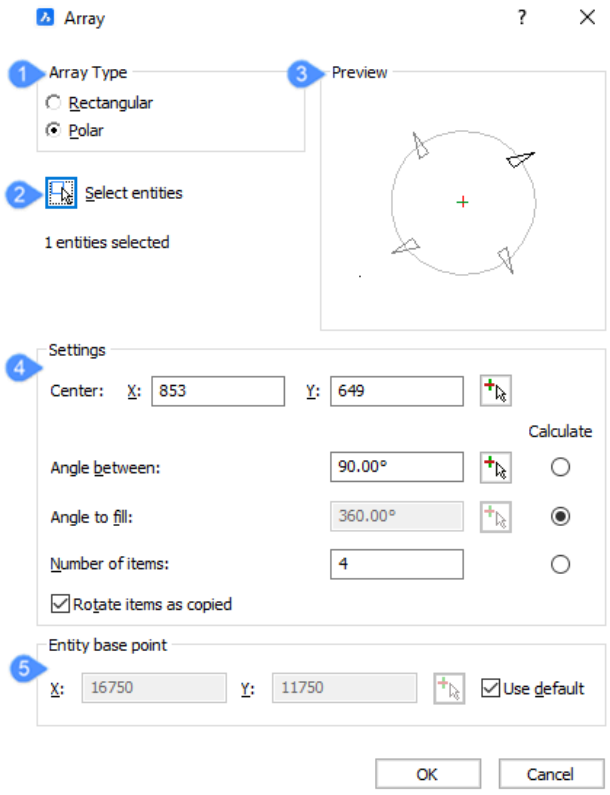
Opens the **Array** dialog box.

Shape Lite Pro Mechanical BIM

6.100.1 Description

Opens the **Array** dialog box.

The **Array** dialog box allows you to create a non-associative 2D rectangular or polar array of entities.



1 Array Type

2 Select entities

3 Preview

4 Settings

5 Entity base point (Polar only)

6.100.2 Array Type

Specifies whether a rectangular or polar array will be created.

6.100.3 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.

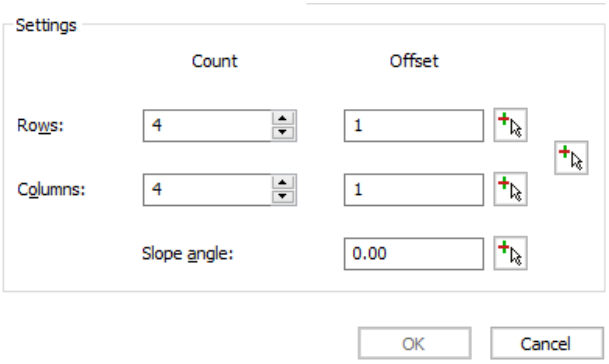
6.100.4 Preview

Displays a preview of the array based on the specified settings.

6.100.5 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 item 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.

6.100.6 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.

6.101 ARRAYCLOSE command

Exits the associative array editing state.

Shape Lite Pro Mechanical BIM

Icon: 

6.101.1 Description

Exits the associative array editing state, displaying the **Array Close** dialog box to save or discard the changes made to the array's source entities.

Note: This command is available only after an associative array is checked out with the ARRAYEDIT command, Source option.

6.101.2 Options within the command

Yes
Saves the changes made to the source entities of the array and then exits the array editing state.

No
Exits the array editing state and discards all changes, thus returning the array to its original state.

Cancel
Closes the dialog box and stays in the array editing state.

6.102 -ARRAYCLOSE command

Exits the associative array editing state.

Shape Lite Pro Mechanical BIM

6.102.1 Description

Exits the associative array editing state, prompting at the Command line to save or discard the changes made to the array's source entities.

Note: This command is available only after an associative array is checked out with ARRAYEDIT command, Source option.

6.102.2 Options within the command

Yes
Saves the changes made to the source entities of the array and then exits the array editing state.

No
Exits the array editing state and discards all changes, thus returning the array to its original state.

6.103 ARRAYDETECT command

Creates arrays of entities (2D or 3D) based on found entity patterns.

Shape Lite Pro Mechanical BIM

6.103.1 Description

Searches for entity patterns in a set of selected entities (2D and/or 3D) and converts them into array entities. Using ARRAYDETECT to replace sets of entities by arrays adds structure to the drawing and reduces file size.

Note: This functionality was extracted from the BLOCKIFY command where it is no longer available.

6.103.2 Methods

There are two methods to detect arrays:

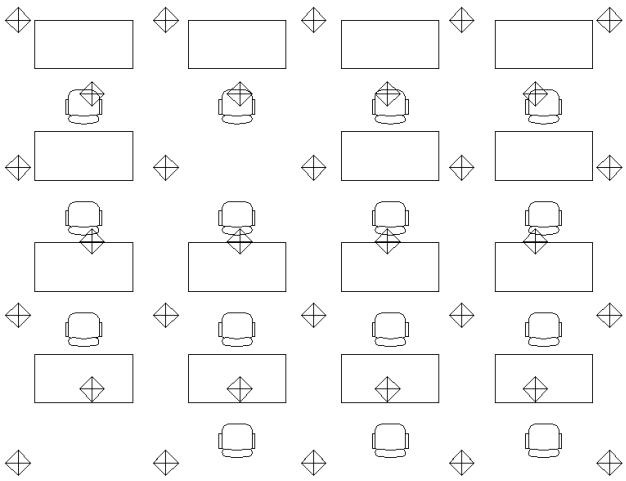
- Multiple entities source.
- Single entity source.

Note: The **Command Context** panel opens to let you choose which of the found patterns using the selected method will be converted to array.

The steps to use ARRAYDETECT command are:

1 Select the entities in which you want to find patterns.

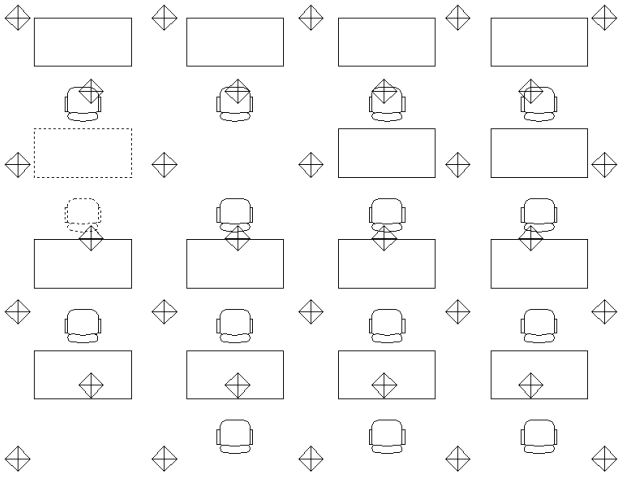
1

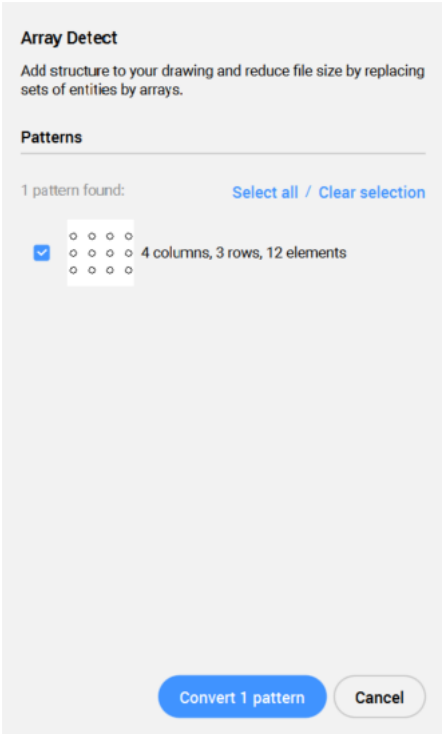


2 Choose one of the two array detection modes:

- Detects patterns with multiple entities source (you have to select the source entities of the patterns to be found).

-

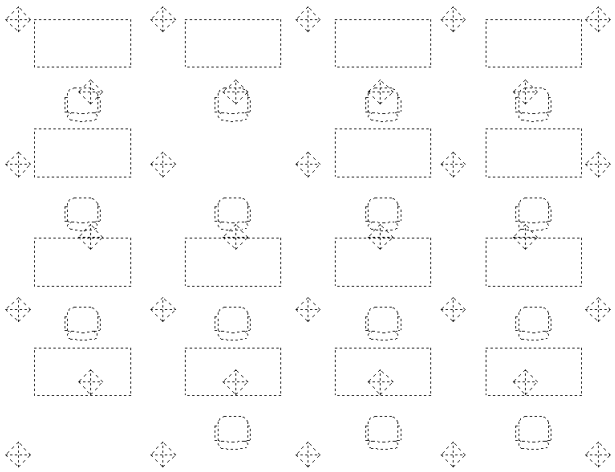


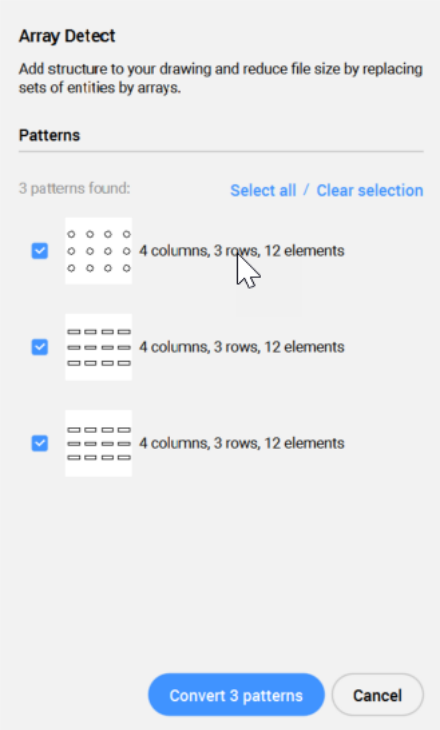


Note: With the **Select all** / **Clear selection** buttons you can select or clear the selection.

- Detects all patterns with a single entity source (no entity selection is needed).

-





Note: To reduce the number of single entity patterns detected, you can use BLOCKIFY command before using ARRAYDETECT command.

3 Specify which of the found patterns need to be converted into arrays.

6.103.3 Options within the command

select source Entities
Detects patterns with the selected multiple entities source.

autodetect Single entity arrays
Detects all patterns with a single entity source.

Convert selected
Converts to arrays all patterns selected in the **Command Context** panel.

convert Separately
Sequentially zooms the view on each found pattern and lets you choose whether to convert it to array.

6.104 ARRAYEDIT command

Edits associative arrays.

Shape Lite Pro Mechanical BIM

Icon: 

6.104.1 Method

Select the entities that make up associative arrays.

6.104.2 Options within the command

Source
Edits the source entities of the array.

Note: Displays **Array Editing State** dialog box. Only the selected entity is displayed in Associative Array Edit environment so it can be easy to modify, add and delete entities. The changes are applied to all the entities once the array editing state is closed.

REPlace
Replace some or all the entities in the array.

RESet
Restores erased entities and removes any item overrides.

Note: Holt the **Ctrl** key to select more than one item in the associative array, then tap the **Delete** key to erase the selection.

eXit
Accept the changes and conclude the command.

6.105 ARRAYEDITEXT command

Provides additional editing of associative arrays.

Shape Lite Pro Mechanical BIM

Icon: 

6.105.1 Description

Provides additional editing for all elements of an associative array or for an individual element in the associative array.

Note: Arrays edited by this command must have been made associative by the ARRAY command. It does not work with arrays created by the ARRAYCLASSIC command or -ARRAY command.

6.105.2 Method

There are two methods when editing an associative array using ARRAYEDITEXT command:

- Edit all elements of the array: interactively adjusts the spacing between them and changes the number of elements.
- Edit an individual item of the array: moves (offset), scales, rotates or deletes it.

6.105.3 Options within the command

SPacing
Adjusts the spacing of an entire associative array interactively.

REsizing
Resizes the entire array by removing entities interactively.

Offset item
Moves the entities of an associative array to a different location.

Note: Pick points or enter x, y coordinates.

Scale item
Resize an entity of an associative array.

Note: A scale factor larger than 1 makes the entity larger and smaller than 1 makes is smaller.

Rotate item
Rotates an entity of an associative array.

Note: Positive numbers, for the rotation angle, rotate counter clockwise and negative numbers rotate clockwise.

Delete item
Deletes an entity of an associative array.

BEdit Source
Opens the source of the array for editing with the BEDIT command.

Note: After erasing the expression, the value is returned to the previous state. Each saving of the BEDIT session is considered as a new state.

6.106 ARRAYPATH command

Creates an array along a path.

Shape Lite Pro Mechanical BIM

Icon: 

6.106.1 Description

Associatively distributes entity copies evenly along a path into multiple rows and levels.

6.106.2 Options within the command

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the Array properties, found in **Mechanical browser**, **Properties** panel or through the ARRAYEDIT command.

Method
Specifies how to distribute entities along the path.

Divide
Distributes a specific number of items evenly spaced along the path.

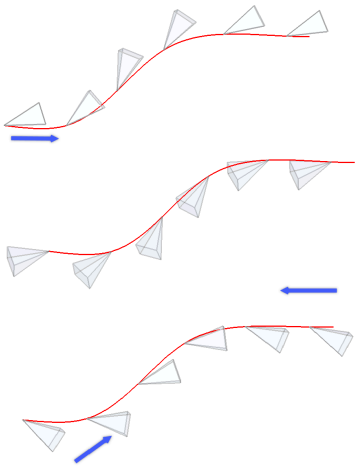
Note: Distributes a specific number of items evenly spaced along the path.

Measure
Places a sufficient number of items along the path, separated by the specified interval.

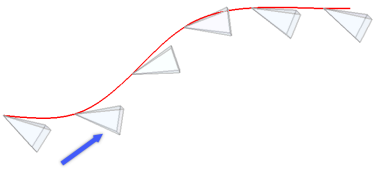
Base point
Sets the base point of the array. Items are placed relative to the base point.

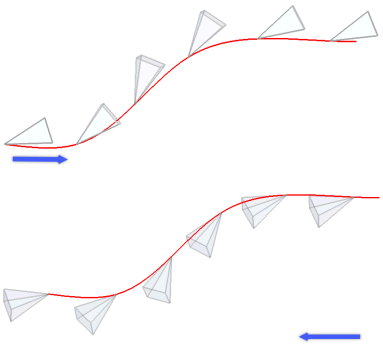
Tangent direction
Determines how the items are aligned relative to the starting direction of the path.

2 Points
Specify two points that define the tangency of the first item relative to the starting direction of the path. Depending on the Align Items setting, the other items are placed parallel to the first item or at the specified tangent direction relative to the direction of the path. In the image **Tangent direction - 2 points**, the blue arrow indicated the tangency direction.



Tangent direction - 2 points





Normal
Orients the Z direction of the item with the starting direction of the path.

Items
Specifies the number of items or the distance between items, depending on the Method setting.

Enter number of items along path
Places items along the entire path at the specified distance.

Note: Placing items along a portion of the path is possible, by specifying a smaller number.

Enter the distance between items along path
Specifies the distance between items by entering a value or picking two points.

Expression
A mathematical formula or equation can be used to derive the value.

Fill entire path
Fills the entire path with items at given spacing.

Note: This option operates like grips editing. If the number of items changes, the length of the path should also change. When the path is modified using grips editing, the array entity is recreated along the modifies path.

Rows
Specifies the number of rows in the array, the distance between them and the incremental elevation.

Distance between rows
Specify the distance between subsequent rows by entering a value or picking two point.

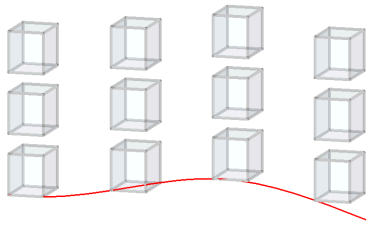
Total
Specify the distance between the first and the last row.

Incrementing elevation between rows
Defines the increasing or decreasing elevation for each subsequent row.

Levels
Creates 3D arrays by specifying the number and spacing of levels.

Distance
Specifies the distance between the bottom and the top level.

Total
Specifies the distance between the bottom and the top level.

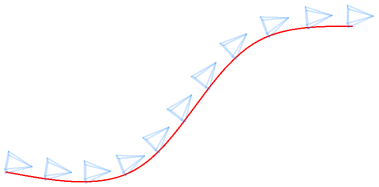


Align items

Specifies whether to align each item tangent to the path's direction. Alignment is relative to the first item's orientation.

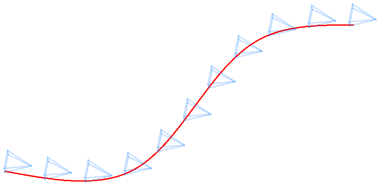
Yes

Each item is placed tangent to the path direction.



No

Each item keeps the orientation of the first item.

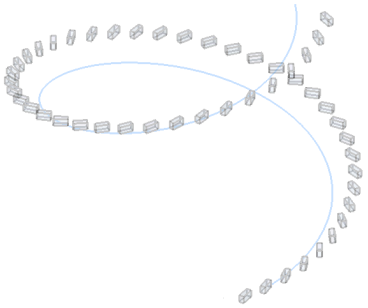


Z direction

Determines whether to maintain the items' original Z direction or to bank the items naturally along a 3D path.

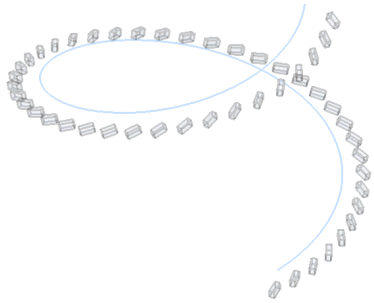
Yes

Z direction (vertical) is maintained.



No

Items banked along the helix.



eXit
Accept the current settings and conclude the command.

6.107 ARRAYPOLAR command

Creates an array of entities, in a circular pattern.

Shape Lite Pro Mechanical BIM

Icon:



6.107.1 Description

Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.

6.107.2 Options within the command

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the **Array** properties, found in **Mechanical browser**, **Properties** panel or through the ARRAYEDIT command.

Base point
Sets the base point of the array. Items are placed relative to the base point.

Items
Specifies the number of items in the array.

Angle between
Specifies the angle between two subsequent items.

Fill angle
Specifies the angle between the first and last item.

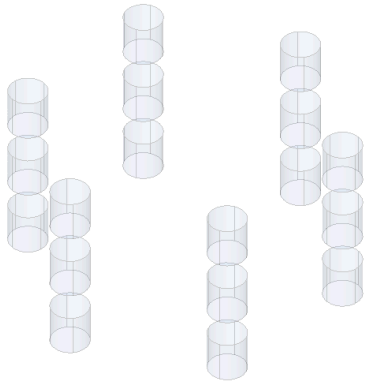
ROWs
Specifies the number of rows, the spacing between subsequent rows and the incremental elevation.

Distance
Specifies the distance between subsequent rows.

Total
Specifies the distance between the first and the last row.

Incremental elevation between rows
Specifies the increasing or decreasing elevation for each subsequent row.

Levels
Creates 3D arrays by specifying the number and spacing of levels.

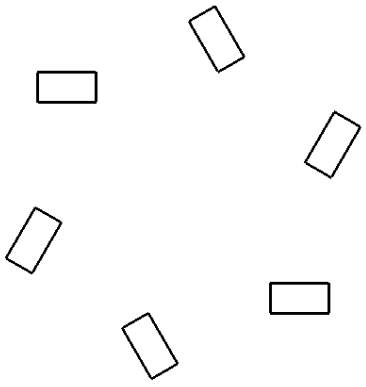


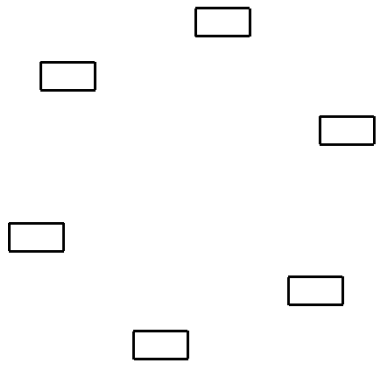
Distance
Specifies the distance between subsequent levels by typing a value or picking two points.

Total
Specifies the distance between the bottom and the top level.

Expression
A mathematical formula or equation can be used to derive the value.

ROTate
Determines whether items are rotated around the circular path or they preserve the orientation of the source entities.





eXit
Accept the current settings and conclude the command.

6.108 ARRAYRECT command

Creates a rectangular array of entities.

Shape Lite Pro Mechanical BIM

Icon:

6.108.1 Description

Associatively distributes copies of entities into any number of rows, columns and levels.

6.108.2 Options within the command

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the Array properties, found in **Mechanical browser**, **Properties** panel or through the ARRAYEDIT command.

Base point
Sets the base point of the array. Items are placed relative to the base point.

COUnt
Specifies the number of rows and columns.

Spacing
Specifies the distance between columns and/or rows.

Unit cell
A rectangle is displayed dynamically. Specify a point or enter the desired column and row spacing in the dynamic entry fields. To switch back and forth between the entry fields, press Tab key.

Note: If dynamic input is on, then the width and height of the spacing rectangle are displayed.

COLumns
Specifies the number of columns and the spacing between subsequent columns.

Expression
A mathematical formula or equation can be used to derive the value.

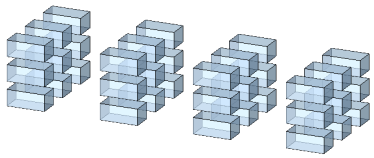
Rows
Specifies the number of rows, the spacing between subsequent rows and the incremental elevation.

Distance
Specifies the distance between subsequent columns/rows.

Total
Specifies the distance between the first and the last column/row.

Incrementing elevation between rows
Specifies the increasing or decreasing elevation for each subsequent row.

Levels
Creates 3D arrays by specifying the number and spacing of levels.



Distance
Specifies the distance between subsequent levels by typing a value or picking two points.

Total
Specifies the distance between the bottom and the top level.

eXit
Accept the current settings and conclude the command.

6.109 ATTACH command

Attaches Xrefs, images, or PDF files in the current drawing.

Shape Lite Pro Mechanical BIM

6.109.1 Method

Note: The ATTACH command unifies the XATTACH, IMAGEATTACH, and PDFATTACH commands.

Opens the **Select Reference File** dialog box that allows you to select the type of the file you want to attach.

Standard Drawing File
After selecting a standard drawing file, click the **Open** button to display the **Attach External Reference** dialog box. See also the **XATTACH command** article.

Image Files
After selecting an image file, click the **Open** button to display the **Attach Raster Image** dialog box. See also the **IMAGEATTACH command** article.

PDF Underlay Files
After selecting a PDF file, click the **Open** button to display the **Attach PDF Underlay** dialog box. See also the **PDFATTACH command** article.

6.110 -ATTACH command

Attaches Xrefs, images, or PDF files in the current drawing via Command line.

Shape Lite Pro Mechanical BIM

6.110.1 Method

Specify the path of the file you want to attach.

Note: The -ATTACH command unifies the -XATTACH, -IMAGEATTACH, and -PDFATTACH commands. See also the related articles.

6.111 ATTACHCIVILOBJECT command

Attaches an external civil object referenced file in the current drawing.

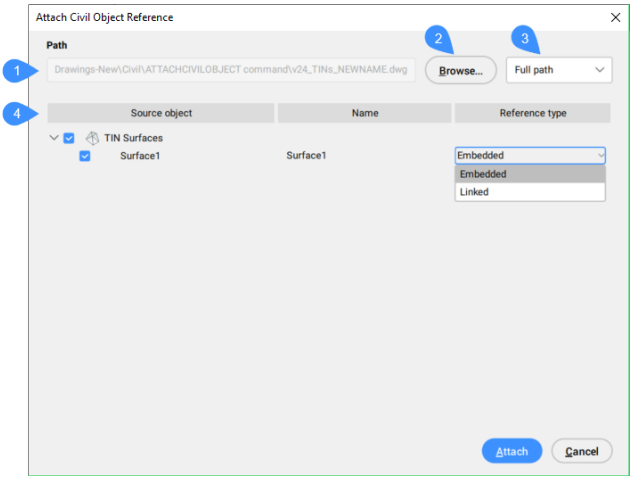
Shape Lite Pro Mechanical BIM

Icon:

6.111.1 Description

Opens the **Attach Civil Object Reference** dialog box to select a DWG file containing a civil object to reference it in the current drawing. After selecting the file and choosing **Open**, the **Attach Civil Object Reference** dialog box displays. The attached civil object (surface) will be visible in the **Civil Explorer** panel.

The **Attach Civil Object Reference** dialog box allows you to attach civil object references to the current drawing.



1 Path

2 Browse

3 Path type

4 Attached civil object

6.111.2 Path

Displays the path of the drawing file.

6.111.3 Browse

Opens the **Open file** dialog box that allows you to choose a different civil object DWG file.

6.111.4 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.

6.111.5 Attached civil object

Lists the properties of the civil object.

Note: These properties are available for editing in the **Info** tab of the **Civil Explorer** panel.

Source object
Displays the source civil object name.

Note: A warning icon displays if the civil object is already referenced in the drawing.

Name
Displays the name of the attached civil object.

Reference type
Defines how the civil object will be attached in the current drawing.

Embedded
The geometry of the reference surface is saved in the drawing. The drawing will be larger but open faster unless the source surface has been changed. Reference surfaces will be visible even if the source drawing is not available.

Linked
The geometry of the reference surface is not saved in the drawing.

6.112 ATTACHMENTSPANELCLOSE command

Closes the **Attachments** panel.

Shape Lite Pro Mechanical BIM

6.112.1 Description

Closes the **Attachments** panel to hide it from the current workspace. If the **Attachments** panel is stacked when you close it, the **Attachments** tab or icon is removed from the stack.

6.113 ATTACHMENTSPANELOPEN command

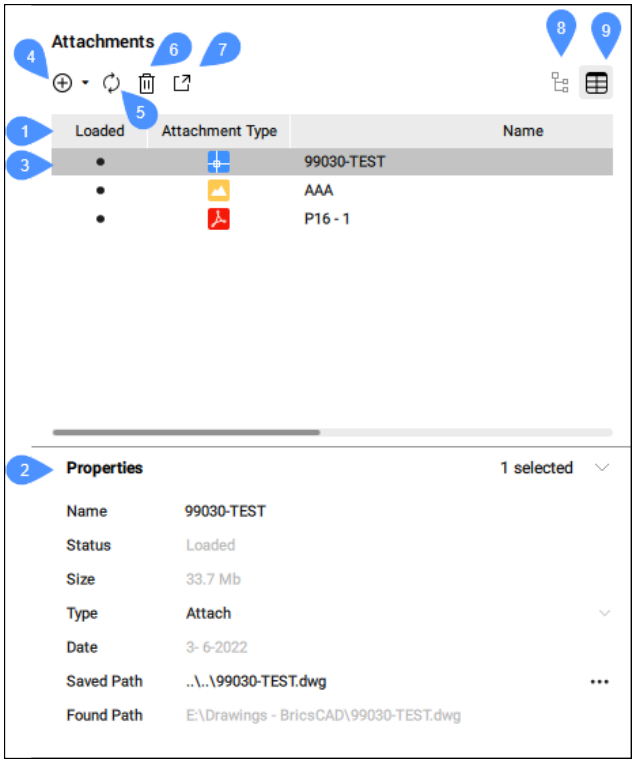
Opens the **Attachments** panel.

Shape Lite Pro Mechanical BIM

6.113.1 Description

Opens the **Attachments** panel to display it in the current workspace. The **Attachments** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Attachments** panel can be either floating, docked or stacked.

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



1 Attachments properties list header

2 Properties

3 Attached files list

4 Attach a file

5 Refresh

6 Detach

7 Open

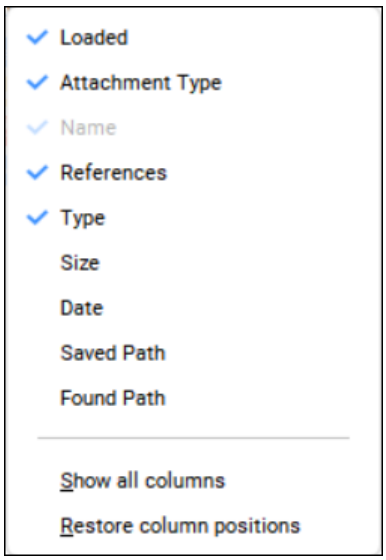
8 Tree view

9 Grid view

6.113.2 Attachments properties list header

Column headers describe theproperties of each attachment. You can sort theattached files list in ascending or descending order foranyproperty by clicking once or twice on thecolumn header. A right-click menuon the column headers enables you to customize them:

Note: Some columns are hidden by default.



Loaded
Specifies the load status of the attachments:

- **Load**: the attachments are loaded and displayed in the drawing editor.
- **Unload**: the attachments are not loaded or displayed in the drawing editor, but maintain their attachment information, such as scale and location.
- **Not found**: the path is not found. The attachments have to be re-attached.

Attachment Type
Displays the attached file type (DWG, PDF, image or point cloud).

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.

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

Note: Only visible block reference inserts on all nested levels are counted.

Type
Specifies the attachment type. Click the attachment type to switch between **Attach** and **Overlay**.

Date
Specifies the date on which the attached files were last saved.

Saved Path
Specifies the path, including the 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.

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

6.113.3 Properties

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

6.113.4 Document name

Displays the active document's name.

6.113.5 Attached files list

Displays a list of attached filesand their properties.

Note: When you select an XREF in the drawing area, it is highlighted in the **Attachment** panel and vice versa.

6.113.6 Attach button

Attach DWG
Displays the **Select Reference File** dialog box like the XATTACH command.

Attach PDF
Displays the **Select PDF Underlay File** dialog box like the PDFATTACH command.

Attach Image
Displays the **Select image file** dialog box like the IMAGEATTACH command.

Attach Point Cloud
Displays the **Point Cloud Reference Manager** dialog box like the POINTCLOUDREFERENCE command.

6.113.7 Refresh

Updates the Grid or Tree Views.

6.113.8 Detach

Detaches the selected attachments from the drawing.

6.113.9 Open

Opens the selected attachments.

6.113.10 Tree View

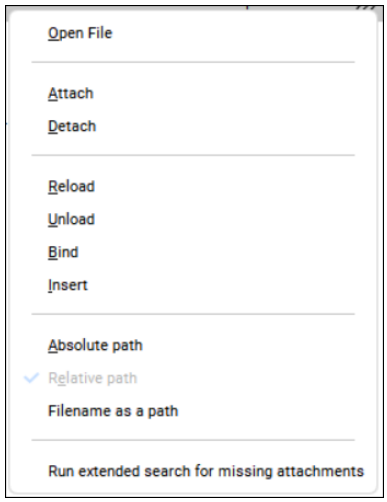
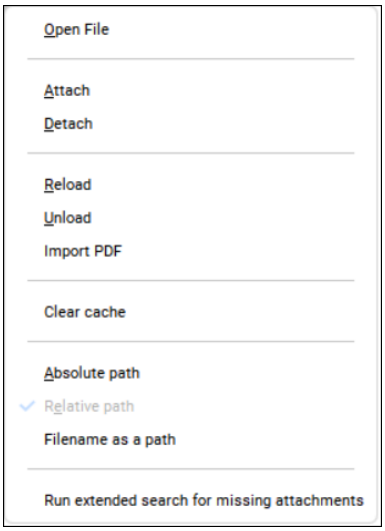
Displays the attachments in a structured tree view.

6.113.11 Grid View

Displays the attachments in a detailed grid view.

6.113.12 Context menu

A right-click menu on the selected attachments offers additional menu options depending on the file type:

DWG file	PDF file
	

Open file
Opens the DWG attachment file.

Attach
Attaches a new reference of the attachment file.

Detach
Detaches all references of the attachment.

Reload
Reloads all references of the attachment, and all its inserts, including parametric ones, are rebuilt.

Unload
Unloads all references of the attachment.

Bind
Copies the contents of the attached drawing to the current drawing as a block reference (-XREF: _B command), according to the value of the BINDTYPE system variable.
Parametric external references are converted to local parametric blocks.

Insert
Inserts all instances of the attachment in the drawing as block references (-XREF: _I command).
Parametric external references are converted to local parametric blocks.

Run extended search for missing attachments
Searches for the attached file in an upper level folder if the folder is added in SRCHPATH system variable.

Import PDF
Launches the -PDFIMPORT command to import the attachment.

Clear cache
Removes the files from the related disk cache folder.

6.114 ATTDEF command

Opens the **Define attribute** dialog box.

Shape Lite Pro Mechanical BIM

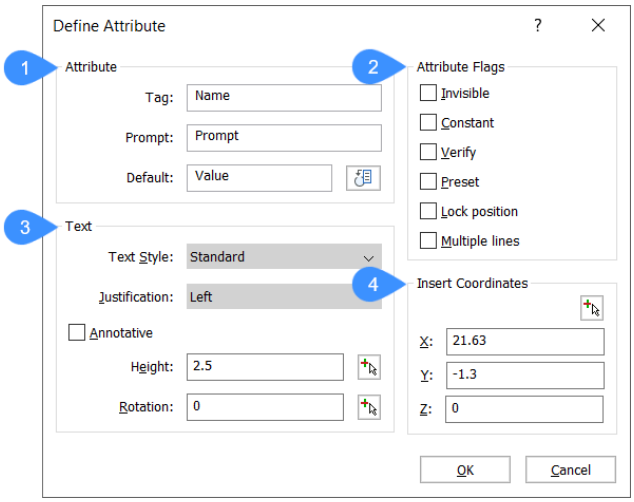
Icon:

Alias: AT, DDATTDEF

6.114.1 Description

Opens the **Define attribute** dialog box to add an attribute to a block definition.

The **Define Attribute** dialog box defines the options for the attribute values.



1 Attribute options

2 Attribute Flags options

3 Text options

4 Insert Coordinates options

6.114.2 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, for example <360>. Press **Enter** to accept the value.

Click the **Insert Field** icon to assign a field value to the attribute (see the FIELD command).

6.114.3 Attribute Flags options

- **Invisible**

  Hide the attribute from view; they are not displayed and not printed. However, hidden attributes can be displayed with the ATTDISP command.

- **Constant**

  Specify a default value that cannot be changed.

- **Verify**

  Forces to enter the value a second time. This helps to ensure that the value is entered correctly.

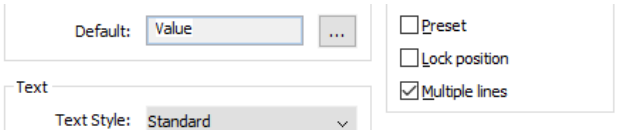
- **Preset**

  Inserts attributes without prompting. The attributes can be changed 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 **Define Attribute** dialog box.

6.114.4 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.

  Note: Supported annotation scales of a selected block attribute can be modified from the **Properties** panel, even when its owning block reference is not annotative.

6.114.5 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.

6.115 -ATTDEF command

Defines attribute data.

Shape Lite Pro Mechanical BIM

Alias: -AT

Note: This command is meant for use by macros, scripts, and LISP routines. To specify attributes with a dialog box, use the ATTDEF command.

6.115.1 Description

Defines attribute data (used by blocks) at the Command line.

6.115.2 Options within the command

Define an attribute by toggling attribute modes.

Toggle attribute modes: Constant = Off Invisible = Off Preset = Off Verify = Off Lock position = Off Annotative = Off Multiple Lines = Off

Constant – type **C**

- Off - users can change the value of the attribute.
- On - users cannot change the value.

Invisible – type **I**

- Off - displays the attribute in the drawing.
- On - hides the attribute

Note: Invisible attributes are not displayed nor printed, but can be displayed with the ATTDISP command.

Preset – type **P**

- Off - prompts the user to enter values.

- On - inserts attributes automatically, without prompting the user.

Note: The value of attributes can be changed later with the ATTEDIT command.

Verify -type **V**

- Off - user enters the attribute value once.
- On- forces the user to enter the value a second time to ensure the value is entered correctly.

Lock position – type **L**

- Off - users can move attributes using grip editing.
- On - locks the position of the attribute within the block reference.

Annotative – type **A**

- Off – creates attributes using a non-annotative style.
- On - creates attributes using an annotative style.

Multiple lines - type **M**

- Off - forces the use only of single-line text.
- On - allows the use uses multiline text.

Tag name of attribute
Specifies the tag of the attribute; this is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers and punctuation.

Prompt
Specifies the user prompt; this is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank; BricsCAD uses the tag as the prompt during attribute insertion.

Default text
Specifies the default value and set the style and justification of the text, as with TEXT command. If the Multiple lines mode is on, you may introduce more text lines.

Text start point
Specify the start point of the text.

use defined Style
Specifies the name of the text style.

Align on line
Aligns the text between two points. The text height is adjusted to maintain its aspect ratio.

Fit between points
Aligns the text between two points.

Center horizontally
Centers the text's baseline on the pick point.

Middle (hor/vert)
Centers the text on the pick point.

Justification options...
Accesses additional justification options.

Height of text
Specifies the height of the attribute text, unless overridden by the text style or justification.

Rotation angle of text
Specifies the rotation angle of the rotation text.

6.116 ATTDISP command

Sets the display mode of attribute text in the drawing.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AD

6.116.1 Description

Toggles the display of attribute text in the drawing, between turn all on, turn all off, or turn on only non-visible ones.

When attributes are off or invisible, they are not displayed in the drawing and are not plotted.

6.116.2 Options within the command

Attribute display mode
Changes the display of attribute values:

- display attribute (**ON**): displays all attributes, including those set to invisible mode by the ATTDEF command.
- hide attributes (**OF**): hides all attributes.
- display **N**ormally: displays attributes, except those set to invisible mode.

6.117 ATTEDIT command

Edits the values and properties of attributes.

Shape Lite Pro Mechanical BIM

Icon:

Alias: ATE

6.117.1 Description

Edits the values and properties of attributes. This command is meant for editing attributes globally with macros, scripts, and LISP routines. For editing individual attributes, it is easier to use the **Attribute Editor** dialog box of the EATTEDIT command.

6.117.2 Methods

There are two methods:

- **Y**es – edit one at a time: edits attributes one at a time; allows changes to values and properties.
- **N**o – edit multiple: edits attributes globally; allows changes to values only.

6.117.3 Options within the command

Edit attributes of which blocks
Specifies the names of blocks; you can use wildcards to specify similarly named blocks: * refers to all characters in the block's name.

Edit attributes with these names
Specifies the names of attributes. You can use wildcards to specify similarly named tags.

Edit attributes with this text
Specifies the values of attributes. You can use wildcards to specify similarly named values.

Select attributes
Chooses the attributes to edit.

Position
Moves the attribute's insertion point to the new specified point.

Angle
Changes the rotation angle of the attribute text to the specified angle.

Text
Changes the value of the attribute.

- **Change**: searches and replaces text.
- **Replace**: new attribute text replaces the text of the currently selected attribute.

Style
Changes the text style of the attribute; the style name must exist in the drawing. (Use the STYLE command to create or edit text styles.)
Enter a distinctive style name, or press Enter to keep the style name.

Color
Changes the color of the attribute text by entering a different color name or number.

Height
Change the height of the attribute text by entering a different text height, pick two points in the drawing, or else press Enter to keep the height.

Layer
Changes the layer of the attribute; the layer name must exist in the drawing. (Use the Layer command to create layers.)
Enter a different layer name, or press Enter to keep the layer name.

Next
Goes to the next attribute; if there are no more attributes to edit, exits the command.

pRevious
Goes to the previous attribute, if any.

Quit
Exits the command.

Edit only attributes visible on screen
This option appears when you choose to edit multiple attributes at a time. It determines which group of attributes to edit:

- **Yes – visible only**: edits only attributes visible in the current viewport.
- **No – edit all**: edits all attributes in the drawing.

Text to change
Specifies the attribute value to change.

New text
Specifies the replacement value.

6.118 ATTEXT command

Opens the **Extract Attributes** dialog box.

Shape Lite Pro Mechanical BIM

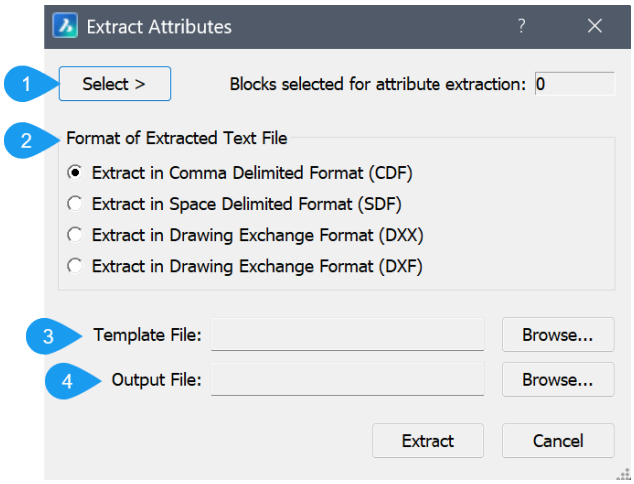
Icon:

Alias: AX, DDATTEXT

6.118.1 Description

Opens the **Extract Attributes** dialog box.

The **Extract Attributes** dialog box allows you to export data from block attributes to a text file.



1 Select

2 Format of Extracted Text File

3 Template File

4 Output File

6.118.2 Select

Specifies the set of blocks with attributes to copy data from.

Note: If blocks without attributes are selected, BricsCAD ignores them.

6.118.3 Format of Extracted Text File

Specifies the format of the extracted text file. Several formats are available:

- CDF: Comma Delimited Format
- SDF: Space Delimited Format
- DXX: Drawing Exchange Format
- DXF: Drawing Exchange Format

6.118.4 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.

6.118.5 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.

6.118.6 Extract

Exports the attribute values to the output file.

6.119 -ATTEXT command

Copies data from attributes to a text file through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -AX

Note: The template TXT file must exist before this command is used. See the ATTEXT command for the format of this file.

6.119.1 Method

There are two methods to use –ATTEXT command:

- Select entities: the **Attribute Extract Template File** dialog box displays.
- Extract as Dxf/Cdf/Sdf: the **Attribute Extract Output File** dialog box displays.

6.119.2 Options within the command

select Entities

Specifies the entities from which will extract attribute value.

Note: If you select blocks without attributes and non-block entities, BricsCAD ignores them. If you ignore this option by entering DXF, SDF, or CDF directly, then BricsCAD selects all attributes in the drawing.

extract as Dxf

Exports the attribute values in a partial DXF format.

Note: This format is meant for use with other computer programs that can read DXX files (DXf eXtraction).

extract as Cdf
Exports the attributes with commas separating values.

Note: This format is meant for use with spreadsheets and databases.

extract as Sdf
Exports the attributes in fixed-width format.

Note: This format is meant for use in word processors.

6.120 ATTIN command (Express Tools)

Inserts block attribute information into the current drawing from an external TXT file.

Shape Lite Pro Mechanical BIM

Icon:

6.120.1 Description

Reads the input .txt file and processes each line. If a block reference with the same handle and block name as the row it is processing is found in the current drawing, it applies the attribute changes to that block.

Note: Make sure the input file has the correct layout, otherwise the ATTIN command will not be able to import the data from the file. It must be in the same format that is generated by the ATTOUT command.

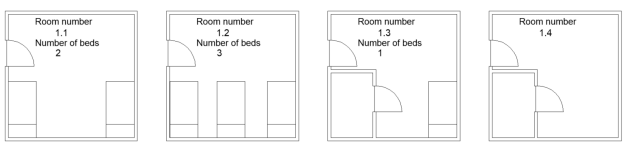
6.120.2 Method

Opens the **Enter file name for attribute input** dialog box that allows you to select an external .txt file.

The file contains information of the block references ordered in a table. The **HANDLE** and **BLOCKNAME** columns give the handle and block name of a block reference. The additional columns give the values of the block attributes. Each row represents a block reference and its values of the attributes. The ATTIN command will search for differences with the current drawing and adapt it according to the imported file.

Note: The string <> indicates that this attribute value does not apply for this specific block.

HANDLE	BLOCKNAME	ROOM_NUMBER	BEDS
'601	Room 4	1.4	<>
'58F	Room 3	1.3	1
'564	Room 2	1.2	3
'53E	Room 1	1.1	2



1 Room number 1.1

Number of beds: 2

2 Room number 1.2

Number of beds: 3

3 Room number 1.3

Number of beds: 1

4 Room number 1.4

6.121 ATTIPEDIT command (Express Tools)

Edits the text of an attribute within a block.

Shape Lite Pro Mechanical BIM

6.121.1 Method

When selecting a single-line attribute, the In-Place Text Editor displays without the **Text Formatting** toolbar and the ruler. Right-click to display the context menu with additional options.

When selecting a multiple-line attribute, the In-Place Text Editor displays with the **Text Formatting** toolbar and the ruler.

6.122 ATTOUT command (Express Tools)

Extracts block attribute information from the current drawing to an external TXT file.

Shape Lite Pro Mechanical BIM

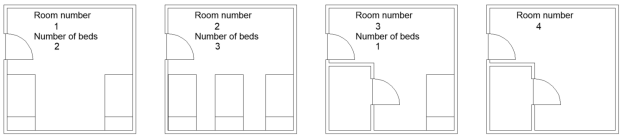
Icon:

6.122.1 Description

Exports data from selected block attributes to a .txt file, which can be reviewed and modified later.

6.122.2 Method

Opens the **Enter file name for attribute output** dialog box that allows you to save the information of the selected block attributes in an external .txt file.



1 Room number 1

Number of beds: 2

2 Room number 2

Number of beds: 3

3 Room number 3

Number of beds: 1

4 Room number 4

The output of the ATTOUT command is a .txt file. The file contains information of the selected blocks ordered in a table. The **HANDLE** and **BLOCKNAME** column give the handle and block name of a block, the

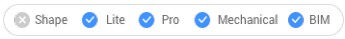
additional columns give the values of the block attributes. Each row represents a block reference and its values of the attributes.

Note: The string **<>** indicates that this attribute does not apply for this specific block.

HANDLE	BLOCKNAME	ROOM_NUMBER	BEDS
'601	Room 4	4	<>
'58F	Room 3	3	1
'564	Room 2	2	3
'53E	Room 1	1	2

6.123 ATTREDEF command

Redefines a block and updates associated attributes.



6.123.1 Description

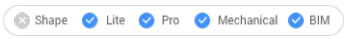
Redefines an existing block reference, and updates any associated attributes, from new or existing entities and attributes available in the drawing.

The following rules apply to existing block references:

- New attributes use default values.
- Values of old attributes that are included in the new block definition are preserved.
- Old attributes that are not included in the new block definition are deleted.
- Format or property changes made with the ATTEDIT or EATTEDIT commands are removed.
- Extended data associated with the block are deleted.

6.124 ATTSYNC command

Synchronizes block attributes.



6.124.1 Description

Synchronizes attribute definitions in all block references of a specified block definition.

6.124.2 Options within the command

Select
Selects the block whose attributes should be used as the template for synchronizing other blocks.

Name
Selects the block definition to be synchronized by name or list of names.

ATTSYNC block?
Synchronizes attributes in the specified blocks.

Yes
Synchronizes attributes in the selected block.

No – skip block
Does not synchronize the block and skips to next block.

6.125 AUDIT command

Analyzes the integrity of the current drawing and fixes errors.

Shape Lite Pro Mechanical BIM

Icon:

Note: Set the AUDITCTL system variable to 1, to let Audit create an ASCII file describing problems and the action taken. This report, with the file extension .adt, is placed in the same directory as the current drawing.

6.125.1 Description

Checks the current drawing for errors and optionally repairs the drawing. It removes duplicate ACIS attributes.

6.125.2 Options within the command

Yes
Fixes errors, if BricsCAD finds any.

No
Searches and reports errors in the drawing database but does not repair them.

6.126 AUTOCOMPLETE command

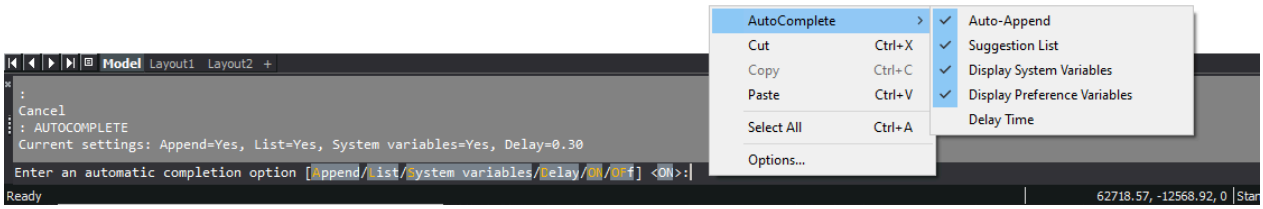
Specifies how the auto-complete function works in the BricsCAD Command line.

Shape Lite Pro Mechanical BIM

6.126.1 Description

This command sets the behavior of the auto-complete function in the BricsCAD Command line. This is done by defining how to append and list which commands and system variables are displayed.

Note: There are six settings, which can also be set by right-clicking in the Command line, using the **AutoComplete** item in the context menu.



6.126.2 Options within the command

Append
Toggles the automatically append suggestion as you type.

List
Toggles the display of the suggestions list.

System Variables
Toggles whether the system variables are included in the list of suggestions or not.

Delay
Specifies the delay time before the list of suggestions displays.

ON
Switches on auto-complete mode.

OFf
Switches off auto-complete mode.

6.126.3 AI-based AutoComplete suggestions

This functionality is an extension to the list of auto-completed commands. This feature adds three more suggestions on top of the ones that are completed in an alphabetical order. These three suggestions are predicted by a Machine Learning model, which is trained based on the commands the user has used in the past.

This feature is available only for the users that choose to be a part of the **Diagnostics & Usage Data Program**.

Note: This feature is set OFF by default. It can be enabled from the **Settings** dialog box.

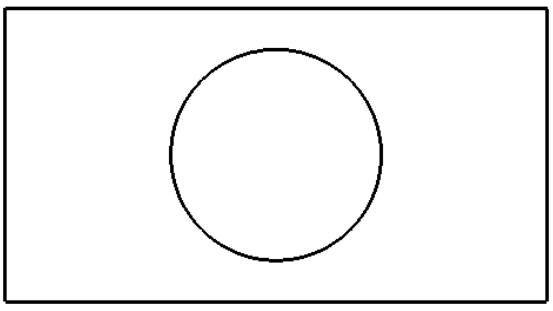
6.127 AUTOCONSTRAIN command

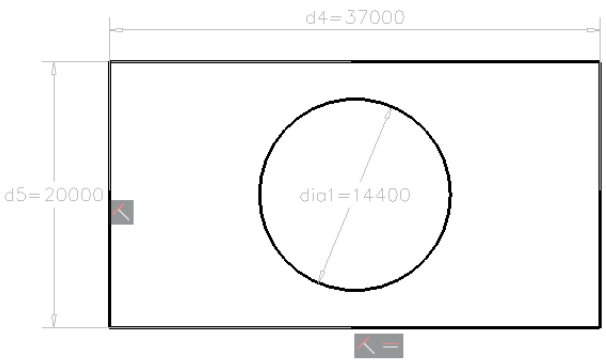
Automatically constrains 2D geometry.

Shape Lite Pro Mechanical BIM

6.127.1 Description

Automatically constrains 2D entities in a drawing, providing both 2D geometric constraints and 2D dimensional constraints.





Note: For BricsCAD Lite license users, constraints can be edited in the **Parameters Manager** panel.

Note: For BricsCAD Pro or higher license users, constraints can be edited in the **Mechanical Browser** panel.

7. B

7.1 BACKGROUND command

Applies a background to the current viewport.

Shape Lite Pro Mechanical BIM

Icon:

Alias: BACKGROUNDS

7.1.1 Description

Opens the **Background** dialog box to apply a background to the current viewport.

You can choose from four types: **None**, **Solid**, **Gradient**, and **Image**.

Note: 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.

7.1.2 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.

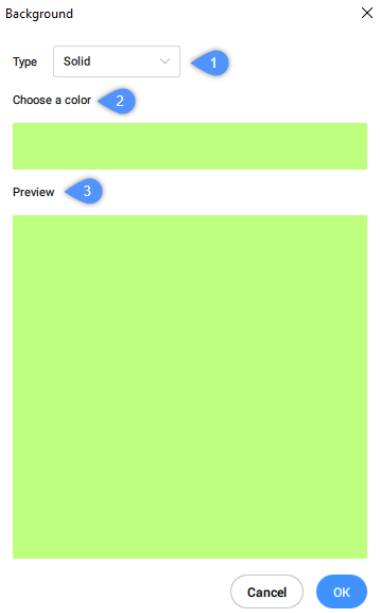
Preview
Displays a preview of the background. When set to **None**, the background depends on the PERSPECTIVE variable:

- **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.

Note: After changing the variables' values, rerun the GRADIENTBKGON command.

7.1.3 Solid

When the background type is set to **Solid**, the **Background** dialog box includes 3 areas:



1 Type

2 Choose a color

3 Preview

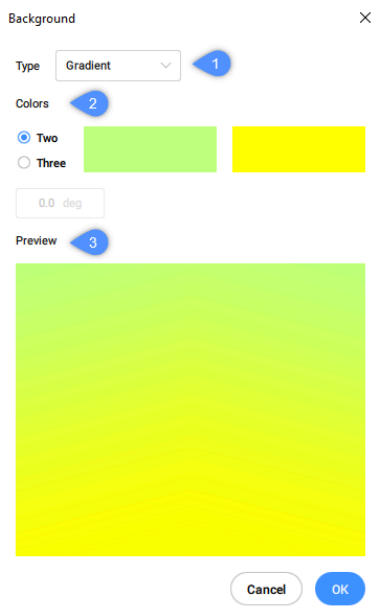
Type
Specifies the background type for the current viewport. When the background type is **Solid**, a single color is applied to the current viewport.

Choose a color
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.

7.1.4 Gradient

When the background type is set to **Gradient**, the **Background** dialog box includes 3 areas:



1 Type

2 Colors

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

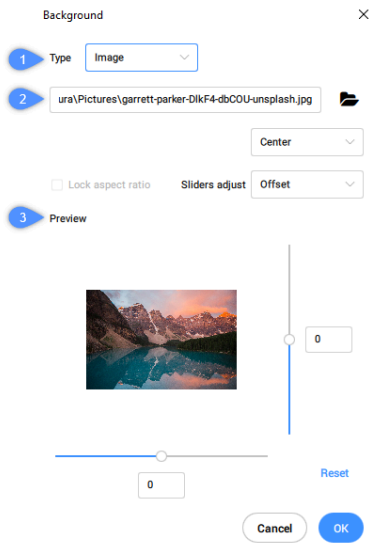
Applies a three-color gradient to the background. When you select this option, the dialog box displays three color swatches. One is for the top color, one is for the middle color, and one is for the bottom color. You can click on the color swatch to display the **Select Color** dialog box and specify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

Preview

Displays a preview of the background using the colors you have selected.

7.1.5 Image

When the background type is set to **Image**, the **Background** dialog box includes 3 areas:



1 Type

2 Choose image and 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.

7.2 BASE command

Changes the drawing's base insertion point.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: BA

7.2.1 Description

Changes the drawing's base insertion point for when it is inserted in other drawings, usually as a block or external reference. The base point can be indicated using X, Y, Z coordinates in the current UCS or by picking a point in the drawing.

7.3 BATTMAN command

Opens the **Block Attribute Manager** dialog box.

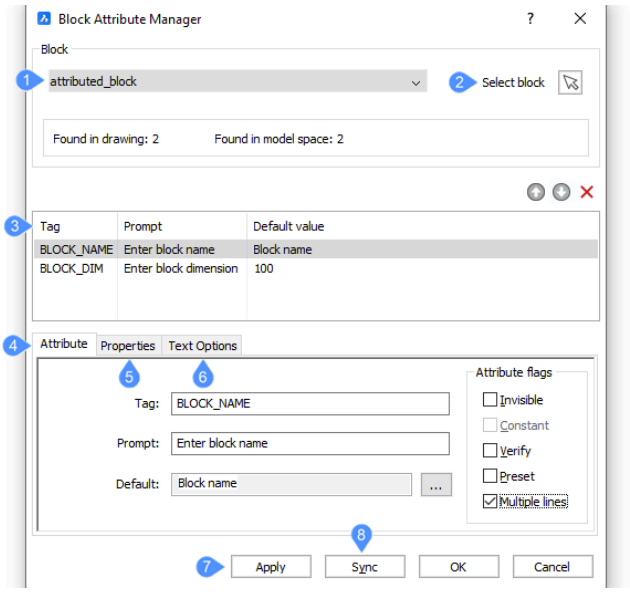
Shape Lite Pro Mechanical BIM

Icon: 

7.3.1 Description

Opens the **Block Attribute Manager** dialog box to view and modify block attribute definitions, properties, and text.

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

7.3.2 Block name

Select a block from the drawing whose attributes you want to edit.

You can select a block from the drop-down list.

7.3.3 Select block

Allows you to select a block in the drawing.

7.3.4 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.

7.3.5 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.

7.3.6 Properties tab

The **Properties** tab allows you to edit the properties of the selected attribute.

7.3.7 Text Options Tab

The **Text Options** tab allows you to edit the text properties of the selected attribute.

7.3.8 Apply

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

7.3.9 Sync

Applies the changes to all instances of the same block in the drawing.

7.4 BCLOSE command

Closes a Block Edit session and saves or discards changes.

Shape Lite Pro Mechanical BIM

Icon: 

7.4.1 Options within the command

Save
Saves changes and closes the **Block Editor**.

Discard block editing changes
Discards changes and closes the **Block Editor**.

7.5 BCOUNT command (Express Tools)

Reports the number of instances of each block in a selection set.

Shape Lite Pro Mechanical BIM

7.5.1 Method

Select entities or press Enter to include all block references. The report is displayed in the Command line.

```
------------------
Table 6p..... 1
Piano........ 1
Sofa......... 3
Sink......... 2
Bed.......... 6
```

7.6 BEDIT command

Opens the **Create or Edit Block Definition** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

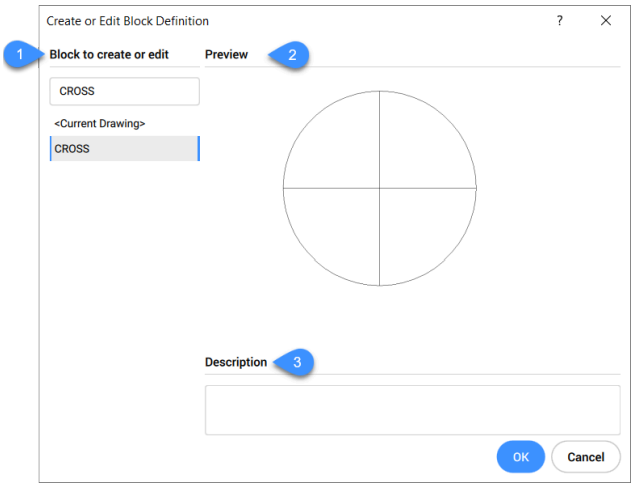
Alias: BE

7.6.1 Description

Opens the **Create or Edit Block Definition** dialog box to specify a block to open in the **Block Editor**.

Note: The HIDEOBJECTS, ISOLATEOBJECTS, and UNISOLATEOBJECTS commands are enabled.

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

7.6.2 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.

7.6.3 Preview

Provides a preview of the selected block.

7.6.4 Description

Displays the description of the selected block.

Click **OK**, to open the **Block Editor**.

7.7 -BEDIT command

Edits the entities that make up a block.

Shape Lite Pro Mechanical BIM

7.7.1 Description

Edits the entities that make up a block in Block Editor environment, after introducing the name of the block to the Command line. Optionally creates a new block.

7.7.2 Method

There are two methods to use -BEDIT command:

- Edit an already existing block.
- Create a new block.

7.7.3 Options within the command

Block name

Enter a block name to open the Block Editor environment and start editing.

Note: Enter an unused name to create a new block.

?

Lists the existing blocks.

7.8 BEXTEND command (Express Tools)

Extends entities to entities nested in blocks and external references.

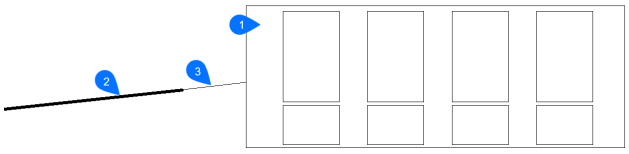
Shape Lite Pro Mechanical BIM

Icon:

7.8.1 Method

Choose one or more entities nested in blocks or external references to use as boundary entity (1). These are the entities to which the subsequently picked entities are extended.

Choose the entity to be extended (2). When another end of an entity could be extended to a boundary, BricsCAD extends the closest end to your pick point.



1 Boundary entity

2 Entity to be extended

3 Extended entity

Note: Switch to trim mode: press and hold the Shift key to select the part of an entity to be trimmed at the intersection with the nearest boundary entities. See the BTRIM command.

7.8.2 Options within the command

Fence
Selects all entities that cross the selection fence. The selection fence is a series of temporary line segments. The selection fence does not form a closed loop.

Crossing
Selects entities within and crossing a rectangular area defined by two points.

Edge mode
Toggles between **Extend** and **No extend**.

Extend
Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.

No extend
Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.

Projection
Specifies the projection method used when extending objects.

No projection
Extends only those entities that intersect real boundaries in 3D space.

xy plane of Ucs
Projects entities and boundaries to the x,y plane of the current UCS, and then extends the projected entities that would intersect projected boundaries.

current View
Projects entities into the current view, and then extends them accordingly.

eRase
Deletes selected entities.

7.9 BHATCH command

Opens the **Hatch and Gradient** dialog box.

Shape Lite Pro Mechanical BIM

7.9.1 Description

Opens the **Hatch and Gradient** dialog box to create a hatch entity in the current drawing.

7.10 -BHATCH command

Fills closed 2D areas with repeating patterns or solid colors.

Shape Lite Pro Mechanical BIM

See -HATCH command.

7.11 BIMADDDETAILREFS command

Creates a link between a master model and a detail file.

Shape Lite Pro Mechanical BIM

7.11.1 Description

The command is used to place detail references in a master model.

- A detail reference is a solid in the model.
- The solid is a **Detail** BIM type, which means that it stores the unique id of a detail in the database.
- The detail reference is a link between a place in the master model and a detail.

Note: The detail reference is placed on the **DETAIL_VOLUME** layer.

7.11.2 Method

Choose a detail to insert as detail reference. Then, choose one of the two modes to add detail references.

7.11.3 Options within the command

Existing solid

You can promote existing solids to detail references by simply clicking on solids. This mode is the easiest one to use, but you must create the solids upfront.

Selection extents

Generates a directed bounding box around (sub)entities selected by the user. You have the option to inflate this box. The generated box will be promoted to a detail reference.

7.12 BIMADDECCENTRICITY command

Controls relative positions of the axis of linear solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.12.1 Description

Adds eccentricity to linear solids with respect to their axis.

7.12.2 Method

Select one or multiple linear solids and choose a base location. You can use the widget or type a location in the Command line.

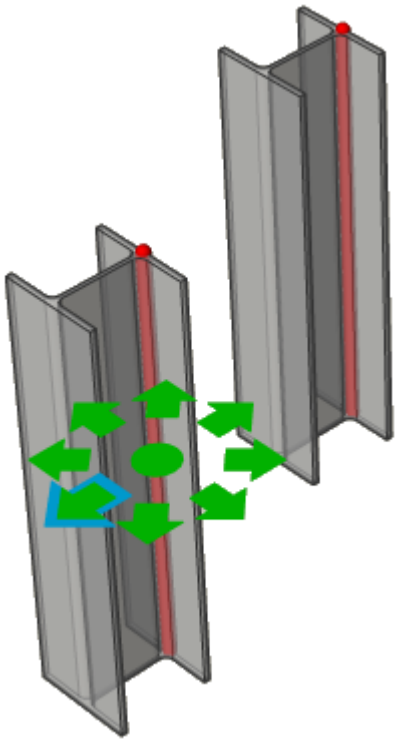
Optionally, you can insert an additional offset vector.

Note: BIMADDECCENTRICITY command only works with structural elements.

7.12.3 Options within the command

Choose base location

- Click one of the nine locations of the widget.
-



- Type a location in the Command line. The options are: **TR:** Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left, **C**: Center (resets the axes of the selection set to the default location)

Extra offset

Allows to move axis to any point.

- **Reset**: resets extra offset.
- **To centroid**: moves axis to the geometrical midpoint of the profile.
- **Eccentricity**: allows to set axis to one of the nine predefined positions.

7.13 BIMALIGNSECTIONBLOCKS command

Aligns section blocks in model space.

Shape Lite Pro Mechanical BIM

7.13.1 Description

This command realigns section results based on *SECTIONRESULTINTERVAL* system variable.

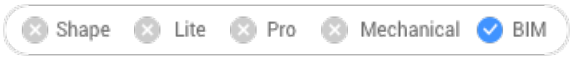
7.13.2 Method

Aligns BIM section blocks in the model space of generated section blocks drawing.

Note: *BIMSECTIONUPDATE* identifies if overlap between generated section blocks happened in model space and suggests calling *BIMALIGNSECTIONBLOCKS* in this case.

7.14 BIMANALYTICALMODEL command

Creates an analytical model from a fully classified BIM model.



7.14.1 Description

Allows you to generate an analytical model from a full 3D model of a structure.

7.14.2 Method

The command opens the **Allowed Deviations From Center Axis** dialog box and starts with an automatically generated proposal but allows users to further adapt the model. For more information, see the related article **Allowed deviations from center axis dialog box**.

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.

Note: The generated proposal can be modified afterward by using the options inside the **Allowed Deviations From Center Axis** dialog box.

7.14.3 Options within the command

Visual settings
Opens the **Visual settings** dialog box. Allows to tick and untick different features of axes and nodes to highlight (in yellow) the axes and nodes that match these features. Also, the size of the disks representing the nodes can be changed from here.

Recalculate
Propagates the effect of a local change further, allowing modifications in the complete model. It will launch a global solution that again tries to minimize rigid links, respecting the changes made.

Export and exit
Exports the model to an IFC (*.ifc) or CIS/2 (*.stp) file.

7.14.4 Options in a node

Select connected axes
Selects the axes connected to the node.

Remove
Removes the node and the connected axes will be disconnected.

Connect from axes
Connects the node to a chosen axis (axes).

Disconnect from axes
Disconnects the node from a chosen axis (axes).

Auto split
Automatically splits the node into two nodes.

Split along axis
Manually splits the node along an axis.

Merge
Merges two or more selected nodes together.

7.14.5 Options in an axis

Select connected nodes
Selects the nodes connected to the axis.

Remove
Removes the axis.

Connect to node
Connects the axis to a chosen node.

Disconnect from node
Disconnects the axis from a chosen node.

Add restrictions
Adds a restriction on the axis.

The restriction you can add depends on the restrictions that are already imposed.

If there are no restrictions yet, you can add the following restrictions:

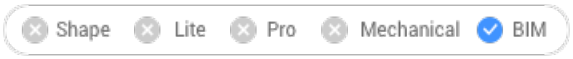
- Parallel to the centerline.
- Inside profile bounds.
- To closest midplane.
- To closest base position.
- To center.
- No length extension.

Relax restrictions
Relaxes a restriction on the axis.
The restriction you can relax depends on the restrictions that are imposed.
Mostly these will be the restrictions that you can relax:

- Inside profile bounds.
- Outside profile bounds.
- Non-parallel.
- Allow length extension.

7.15 BIMANCHOR command (Experimental)

Anchors/un-anchors a block to/from a face of a 3D solid.



Icon:

7.15.1 Disclaimer

EXP.

BIMANCHOR is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

7.15.2 Method

To anchor a block on a 3D solid face, select the block reference, then select a point on a face of a 3D solid. The anchor links the block to the face. The block will stay on the face when the host solid is moved or edited.

To un-anchor a block from a 3D solid face, select the anchored block, then select **Yes**.

7.16 BIMAPPLYPROFILE command

Applies a profile to linear entities and linear solids.

Shape Lite Pro Mechanical BIM

Icon:

7.16.1 Description

Allows you to add profiles, to detach current profile and to convert solids to line.

Note:

- Make sure the selected paths are placed in the XY plane of the current UCS.
- The linear entities accepted as paths are: lines, polylines, arcs, circles, elliptical arcs, ellipses, and helixes.
- Open and closed splines are accepted as paths only if they have default tangents that are not self-intersecting.

7.16.2 Methods

There are two methods to apply profiles:

- Choose a profile from the library.
- Choose a profile entity in the drawing.

7.16.3 Options within the command

Select path

Select linear entities or linear solids.

Select profile
Select a profile entity in the drawing (a closed 2D entity, a region or an existing linear solid) or press **Enter** to display the **Profiles** dialog box.

Apply profile?
- **Quarter turn**: rotates the profile 90° counterclockwise.
- **Rotate**: rotates the profile at a user-defined angle. Positive values rotate the profile counterclockwise.
- **Delete clippings**: removes all clippings from the selected linear solid(s).

LIbrary
Opens the **Profiles** dialog box to change the profile.

convert solids to Line
Converts linear solids to their axis line.

DEtach current profile(s)
Detaches profiles only if the entities have an attached profile.

7.17 BIMATTACHCOMPOSITION command

Attaches/detaches a BIM composition to solids.

Shape Lite Pro Mechanical BIM

Icon:

7.17.1 Method

The command:

- Attaches a selected composition to selected solids.
- Detaches compositions from the selected solids.

Note: To preview in the drawing the attached compositions, turn On the RENDERCOMPOSITIONMATERIAL system variable.

Select a composition and the solids to attach the composition to, then choose a reference face.

The command finds the broken connections (mitered/butted) between the selected solids and adjacent solids and automatically restores them.

Note: Make sure the KEEPCONNECTIONS system variable is turned On.

Accept/reject all restored connections or check connections individually.

There are three methods to select the composition to be attached to solids:

- Enter the composition name.
- Use the **Compositions** dialog box.
- Select an entity with a composition attached.

Note: You can also drag an drop the composition from the **Compositions** panel.

7.17.2 Options within the command

Dialog
Opens the **Compositions** dialog box.
For more information about the dialog box, see the article **BLCOMPOSITIONS command**.

Entity
Applies the composition of an existing solid. First, select the entities to copy the composition from; next, select one or more entities to which to attach the composition.

deTach
Allows you to detach a composition, while keeping the BIM classification, as different from the **Unclassify** option of the BIMCLASSIFY command, which removes all BIM data.

Accept
Accepts all automatically restored wall-wall connections.

Reject
Rejects all automatically restored wall-wall connections.

Check individually
Zooms in the view to each restored connection and lets you accept/reject it.

Yes
Accepts the restored connection. Select **Confirm**.

No
Rejects the restored connection. Select **Confirm**.

Confirm
Confirms the previous option selection.

7.18 BIMATTACHSPATIALLOCATION command

Attaches or removes a spatial location to/from a selection of entities.

Shape Lite Pro Mechanical BIM

Icon: 

7.18.1 Method

Run the command and choose one of the location numbers or choose to attach or unattach locations.

7.18.2 Options within the command

Auto attach locations
Allows you to automatically assign a spatial location. If necessary, new building(s) and stories are created in the **Spatial Locations Manager** dialog box.

Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.

- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

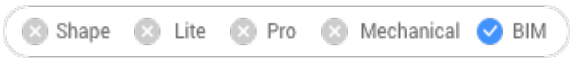
Unattach current location
Allows you to remove the current spatial location from a selection set (including the **Space** property).

Select space
Allows you to assign a space to a selection of entities by first selecting the entities and then the space to be assigned.

Note: Assigning a **Space** property to a building element overrides the element's **Story** and **Building** properties with those of the space.

7.19 BIMAUTOMATCH command

Matches the composition, properties or parameters of example entities to all similar entities in your project.



Icon: 

7.19.1 Method

BIMAUTOMATCH finds similarities based on the type of element, interior/exterior property, orientation, spatial location, amount of windows in a wall and already defined properties. It will not overwrite already defined properties, unless no unmodified entity can be found and the user specifies a source entity.

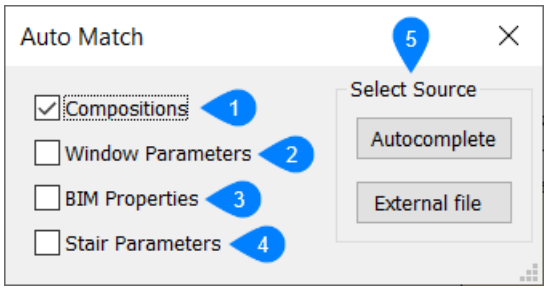
For example: You have attached a specific composition to a wall and a floorslab in the project. Instead of attaching the composition to all the similar walls and floor slabs 1 by 1, you can use this command to do it for you.

Note: This command works best when the project is bimified (see **BIMIFY command** article).

The command displays the **Auto Match** dialog box (see **Auto Match dialog box** article):

Note: When there are unmodified or unclassified entities in the drawing and there is only one building, nothing is prompted.

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

7.19.2 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.

7.19.3 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.

Note: Some window parameters will never be copied along.

- W (= width of the window)
- H (= height of the window)

7.19.4 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

7.19.5 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.

Note: Some stair parameters will never be copied along:

- Height_Staircase
- Stair_Width
- Length_Staircase

7.19.6 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.

7.19.7 Options within the command line

Autocomplete
When choosing **Autocomplete** with multiple buildings in the drawing, you have to specify the building/(s) to apply style on:

All
All the buildings will be taken into account while doing AutoMatch.

Building
AutoMatch is only applied as if **Building** was the only building in the drawing.

0 for building-2
AutoMatch is only applied as if **building-2** was the only building in the drawing.

1 for building-3
AutoMatch is only applied as if **building-3** was the only building in the drawing.

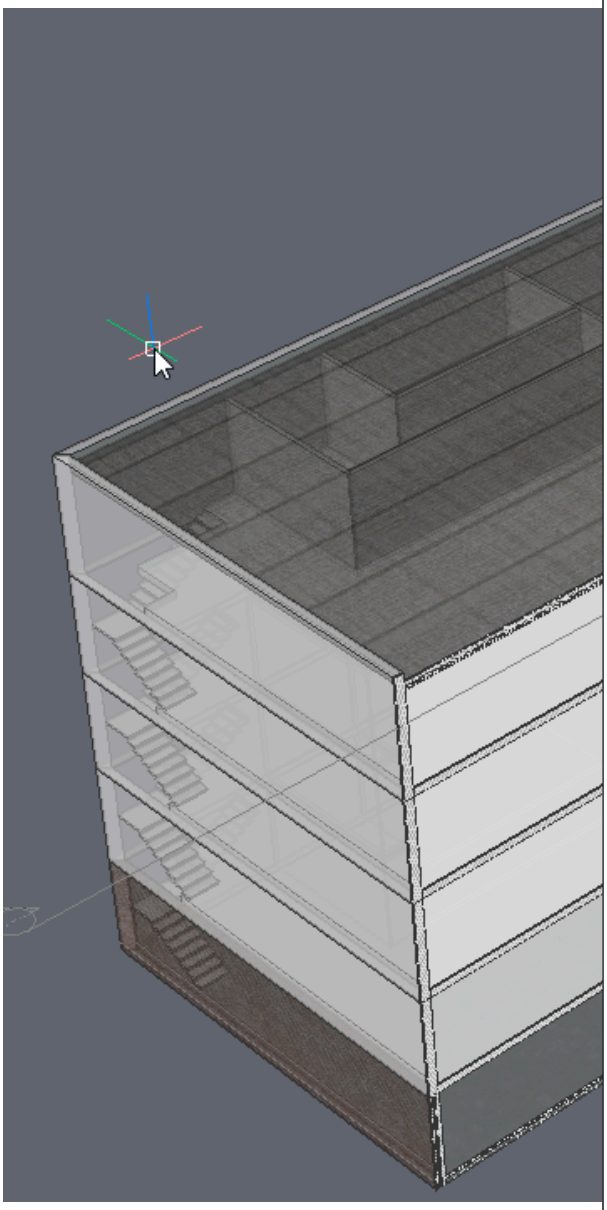
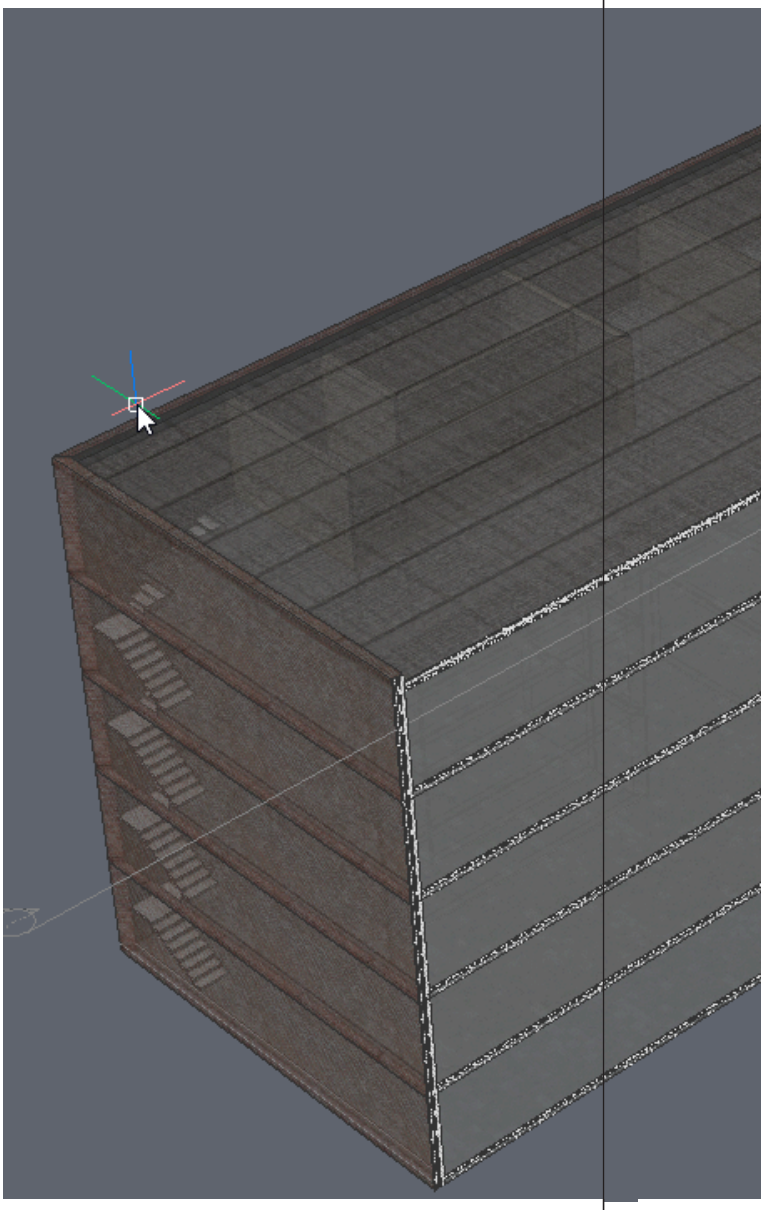
Other building name (first letter capitalized)
All building names, available in the current drawing, are listed here.

Note: AutoMatch is only applied as if **Other building name** was the only building in the drawing.

External file
Matches all chosen options for entities in the external file with similar entities in the open drawing.

Note: When choosing **External file** with multiple buildings in both drawings, you have to select a building to detect style from and all or only one building to apply style on (see above options).

Note: When all entities have compositions or the same customized property or all windows or stairs have customized parameters, you are prompted to select an entity, a window, or a stair in the drawing from which you want to copy the composition or parameters to let AutoMatch overwrite them.

Every type of element has one entity with the correct composition.	All elements are given the correct composition.
	

7.20 BIMBEAM command

Creates solids classified as **Beam**.

Shape Lite Pro Mechanical BIM

Icon:

7.20.1 Description

The command creates beams with different shapes. You can define the options through the command context panel, as well as through the Command line.

7.20.2 Method

Launch the command to open the **Beam** command context panel.

Note: To place beams more easily, enable **Top View Mode** (**TVM**) before launching the command by clicking a story disk ( ) in the **Story Bar** (see **The Story Bar** article). Make sure the top section plane is placed above the beam.

There are two methods to create beams:

- Place single beams, constrained by the X-/Y- axis
- Draw poly beams, unconstrained by the X-/Y- axis

Use the dynamic dimensions to define the insertion point of a single beam more accurately. These dimensions show the distances from the single beam to walls and/or beams. Tap the **TAB** key to switch between dimensions and set them manually.

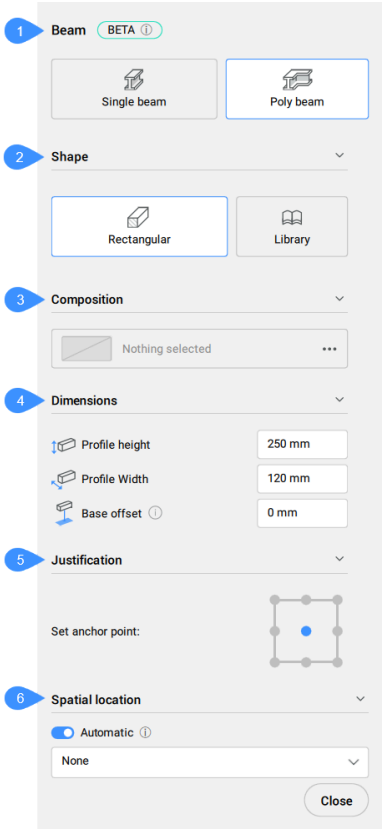
Note: Dynamic dimensions are displayed if **Dynamic Input** (**DYN**) is set to **On** (see **Dynamic dimensions** article).

Use the **Hotkey Assistant** widget to change the direction of the current single beam. Press the **Ctrl** key to switch between options **X-aligned beam** and **Y-aligned beam**.



Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMBEAM options** check box is ticked in the **Hotkey Assistant Configuration** dialog box (see **Hotkey Assistant widget** article).

7.20.3 Options within the command context panel



1 Creation mode

2 Shape

3 Composition

4 Dimensions

5 Justification

6 Spatial location

Creation mode

Choose a method for creating the current beam/beams.

Single beam

Creates X-aligned or Y-aligned single beams.

Poly beam

Creates polyline shaped beams, unconstrained by the X- or Y- axis. You can define the new beams by manually drawing a polyline.

Shape

Defines the current beam profile. You can select an exiting profile or create a new one.

Rectangular

Creates a beam with a rectangular profile. Specific dimension settings are displayed for this profile.

Library

Opens the **Profiles** dialog box to let you select an existing profile or define a new one.

Composition

Click the browse button (...) to open the **Compositions** dialog box that lets you define the current beam composition. Here you can change the composition type by selecting a new filter from the top left drop-down menu.

Note: Beam profiles require variable ply thickness. A warning icon is displayed in the **Composition** section when you select a composition with a fixed thickness.

Dimensions

Profile height/Profile width

Sets the profile height/width.

Note: These options are only available for the **Rectangular** profile.

Base offset

Sets the offset value for the beam base.

Note: A base offset may result in beams being created above the top clipping plane of the **TVM** making them invisible. To avoid this, drag the top clipping plane to a higher elevation in the **Story Bar**.

Justification

There are nine justification points: top left, top center, top right, middle left, middle center, middle right, bottom left, bottom center, bottom right. By default, the anchor point is set to middle center. To change it, click another displayed anchor point.

Spatial location

Allows you to select a spatial location from the drop-down menu to assign to the beam.

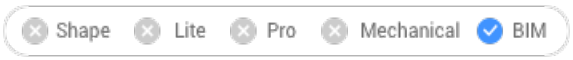
Automatic

Copies the spatial location of the closest underlying slab or in **TVM** the active story.

Note: The options within the command context panel and those within the **Hotkey Assistant** widget reflect the options within the Command line.

7.21 BIMBUILDINGSYSTEM command

Creates or edits a BIM building system group.



7.21.1 Method

Enter an existing Building System group name or a name for a new Building System group, then select the entities to add to or remove from the Building System group.

Note: Building Systems created with BIMBUILDINGSYSTEM command are exported with IFCEXPORT command as IfcBuildingSystem.

7.21.2 Options within the command

List building systems

Lists all existing BIM Building System groups in the Command line.

Create a building system
Creates a new BIM Building System group.

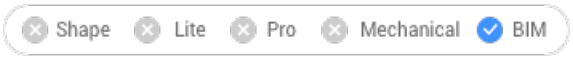
Show properties
Allows you to edit the properties. The available properties are displayed in the Command line.

Ifc properties
Allows you to edit the IFC Properties. The available IFC properties are displayed in the Command line.

Delete a building system
Deletes the selected BIM Building System group.

7.22 BIMCHECKDETAILS command

Inspects the BIM Project details and their references.



7.22.1 Description

The command opens a command context panel that lets you see which BIM Project detail has references in a Project's 3D model/sheet (lists its references, if any). It also checks the links between all detail references/layouts and their DWG files.

7.22.2 Method

Use the command with a Project's 3D model or sheet file open.

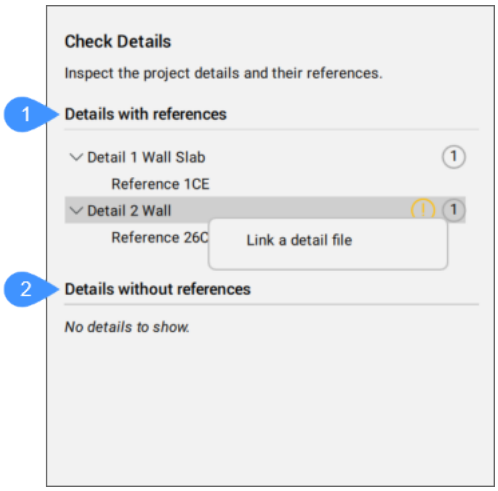
The command context panel displays the details in tree views, with each node representing a detail. For each detail, the number of the detected references is displayed near the detail's name. Expand a node to see the references.

Through the command context panel you can:

- Identify detail references by highlighting in the drawing the ones selected in the panel (for 3D models/sheets).
- Update links between detail references in the model or detail layouts on sheets and detail DWG files (for 3D models/sheets).
- Insert on the current sheet layouts of BIM details referenced in the model (for sheets).
- Update detail layouts inserted on the current sheet (for sheets).

Right-click a detail to display a context menu with actions specific to the current file.

3D model:



1 Details with references

2 Details without references

Details with references
Lists all project details that have references in the 3D model.

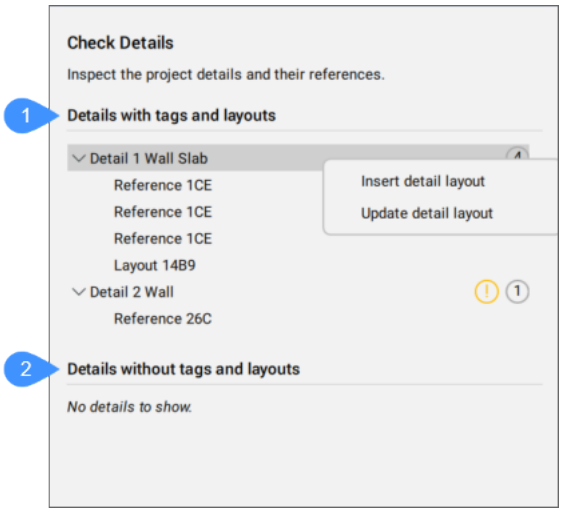
Details without references
Lists all project details without references in the 3D model.

Context menu options:

Link a detail file
Allows you to reassign a detail file to a detail in the database.

Note: The option is available only when the detail file is missing.

Sheet:



1 Details with tags and layouts

2 Details without tags and layouts

Details with tags and layouts
Lists all details on the sheet that have tags and layouts. Expand a node to see the detail references/layouts.

Details without tags and layouts
Lists all details on the sheet that do not have tags and layouts.

Context menu options:

Link a detail file
Allows you to reassign a detail file to a detail/a detail layout in the database.

Note: The option is available only when the detail file is missing.

Insert detail layout
Lists the names of the layouts in the selected detail's file and allows you to insert one the sheet.
Type in a layout name, press **Enter**, then select the insertion's base point and second point.

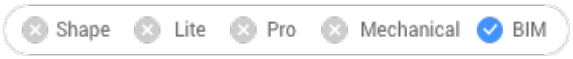
Update layout
Updates the inserted detail layout.

7.23 BIMCLASSIFY command

Classifies an entity and gets a name and an internal unique **GUID** (= globally unique identifier).

Note: The **GUID** is maintained even when the BIM classification changes.

A classification can be assigned to any .dwg entity.



Icons: 

7.23.1 Method

Select the entities to classify or press **Enter** to classify the entire drawing.

The BIM elements definition are detailed in the **Classify As BIM Element** dialog box. For more information, see the related article **Classify As Bim Element dialog box**.

7.23.2 Options within the command

Wall
Classifies the selection as walls.

Note: Attaching a Wall composition to a solid automatically classifies the solid as a wall.

Column
Classifies the selection as columns.

Slab
Classifies the selection as slabs.

Note: Attaching a Slab composition to a solid automatically classifies the solid as a slab.

Beam
Classifies the selection as beams.

wIndow
Classifies the selection as windows.

Door
Classifies the selection as doors.

sPace
Classifies 2D and 3D entities as spaces. (IFC Spatial Structure Element)
Converts BIM rooms to BIM spaces.

buiLding
Classifies 3D entities as buildings. (IFC Spatial Structure Element)

sTory
Classifies 3D entities as stories. (IFC Spatial Structure Element)

building Element
Classifies the selection as unspecified building elements.

Xref
Classifies the selection as External References.

Other
Allows to classify the selection or convert the selection to a block and classify the block reference.

Note: Displays the **Classify As Bim Element** dialog box.

Auto
Automatically classifies the selected entity or entities.

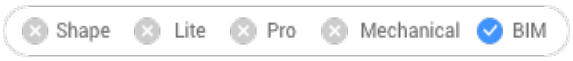
Note: This is the same as running BIMIFY command.

stRuctural
Classifies the selection as Building Structure Elements.

Unclassify
Removes all BIM data from the selected entities.

7.24 BIMCOLLECTDETAILS command

Collects all tagged BIM details from the active sheet and inserts layouts of these details on the sheet.



7.24.1 Method

After it collects all tagged BIM details from the active sheet, it inserts layouts of these details on the sheet. For details with multiple layouts, name a layout to be inserted. Specify the base and second insertion points for the layout.

The inserted layout will hold a pointer to the detail source. This additional information allows BIMCHECKDETAILS command to list and interact with the inserted layouts of the project details.

Note: This command is available only in a BIM Project sheet.

7.25 -BIMCOLLECTDETAILSFROMSHEETS command

Collects all BIM details from named sheets.

Shape Lite Pro Mechanical BIM

7.25.1 Description

Collects all the BIM details from the named BIM Project sheets and inserts layouts of these details as separate block references on the opened sheet.

Note: This command is available only in a BIM Project sheet.

7.25.2 Method

Name the BIM Project sheets one by one from which to collect all the details. Press **Enter** to finish the sheets selection process. All detected details are listed in the upper field of the Command line.

For details with multiple layouts, name a layout to be inserted.

Specify the base and second insertion points for the collection of layouts.

The inserted layouts hold a pointer to the sources of the details. This additional information allows the BIMCHECKDETAILS command to list and interact with the inserted layouts of the Project details.

7.26 BIMCOLUMN command

Creates solids classified as **Column**.

Shape Lite Pro Mechanical BIM

Icon: 

7.26.1 Description

The command creates columns with different shapes. You can define the options through the command context panel that opens when the command is launched, as well as through the Command line.

7.26.2 Method

Launch the command to open the **Column** command context panel. Set the current column options and place the column into the model.

Note: To place columns more easily, enable **Top View Mode** (TVM) before launching the command by clicking a story disk () in the **Story Bar** (see **The Story Bar** article).

Use the dynamic dimensions to define the insertion point more accurately. These dimensions show the distances from the anchor point to other walls and/or columns. Tap the **TAB** key to switch between dimensions and set them manually.

Note: Dynamic dimensions are displayed if **Dynamic Input** (**DYN**) is set to **On** (see **Dynamic dimensions** article).

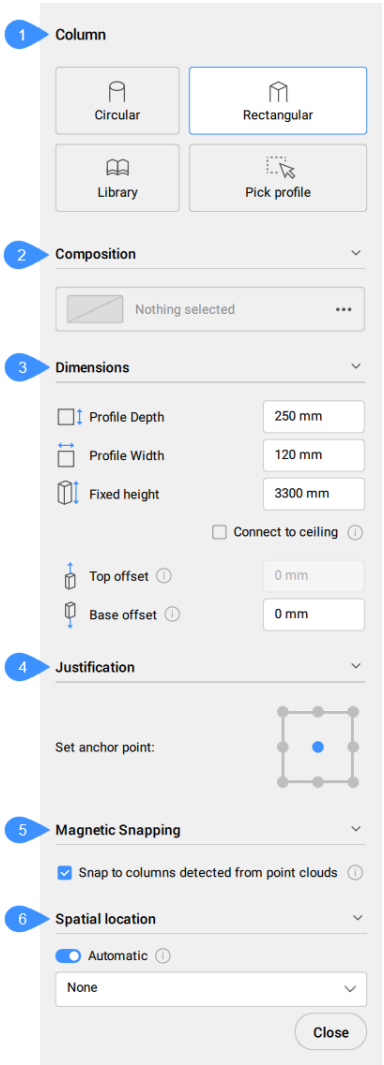
Use the **Hotkey Assistant** widget to rotate the orientation of the profile's cross-section. Press the **Ctrl** key to rotate it by 90 degrees CCW.



Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMCOLUMN options** check box is ticked in the **Hotkey Assistant Configuration** dialog box (see **Hotkey Assistant widget** article).

Use the selection dimensions to reposition the selected column relative to the nearest detected walls and/or columns or other convenient reference points (see **Selection dimensions** article).

7.26.3 Options within the command context panel



1 Column

2 Composition

3 Dimensions

4 Justification

5 Magnetic Snapping

6 Spatial location

Column
Defines the current profile. You can choose an exiting profile, pick one from the drawing or create a new one.

Circular
Creates a column with a circular profile. Specific dimension settings are displayed for this profile.

Rectangular
Creates a column with a rectangular profile. Specific dimension settings are displayed for this profile.

Library
Opens the **Profiles** dialog box to let you select an existing profile or define a new one.

Pick a profile
Allows you to select a profile from the drawing and set it as the current profile.

Composition

Click the browse button (…) to open the **Compositions** dialog box that lets you define the current column composition. Here you can change the composition type by selecting a new filter from the top left drop-down menu.

Note: Column profiles require variable ply thickness. A warning icon is displayed in the **Composition** section when you select a composition with a fixed thickness.

Dimensions

Profile Depth/Profile Width
Sets the profile depth/width.

Note: These options are only available for the **Rectangular** profile.

Profile Radius
Sets the profile radius.

Note: This option is only available for the **Circular** profile.

Fixed height
Sets a fixed height for the column.

Connect to ceiling
Searches for solids above to connect to.

Top offset
Sets the offset value for the top of the column.

Base offset
Sets the offset value for the base of the column.

Justification
There are nine justification options: top left, top center, top right, middle left, middle center, middle right, bottom left, bottom center, bottom right. By default, the anchor point is set to middle center. To change it, click another displayed anchor point.

Magnetic Snapping

Snap to columns detected from point clouds
Toggles **On/Off** to control whether the profile of the column should be adopted from nearby columns detected in point clouds.

Note: This option only works for the **Circular** and **Rectangular** profiles.

Spatial location
Allows you to select a spatial location from the drop-down menu to assign to the column.

Automatic
Copies the spatial location of the closest underlying slab.

Note: The options within the command context panel and those within the **Hotkey Assistant** widget reflect the options within the Command line.

7.27 BIMCOPY command

Creates copies from the planar faces of 3D solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.27.1 Description

Creates copies from the planar faces of 3D solids, and the sides and ends of linear solids.

7.27.2 Method

Specify a distance to create copy.

Note: You can specify a distance in the dynamic dimension field and right-click to create a single copy.

7.27.3 Options within the command

Copy
Creates copies.

Repeat
Repeats the copy using the same displacement.

Note: Move the cursor in the displacement direction. The further you move away from the source solid, the more copies are created.

Number
Creates a number of copies you want to have in your model.

Accept
Accepts the current distance you entered in the dynamic dimension field.

7.28 BIMCREATEDETAIL command

Creates a 3D detail and saves it in the **Details** library.

Shape Lite Pro Mechanical BIM

Icon:

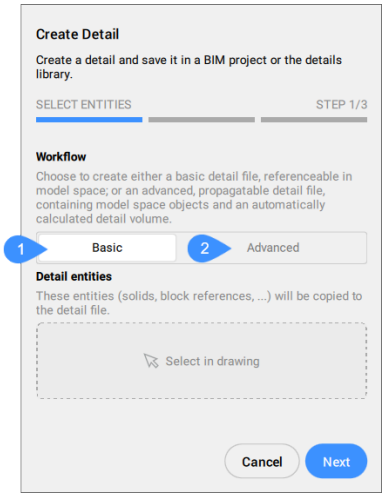
7.28.1 Method

Launch the command to open the **Create Detail** command context panel that allows you to define a BIM 3D detail in three steps.

Note: The options within the Command line reflect the options found in the command context panel.

You can choose to create:

- A **basic** detail - can be saved to the **Project** and can be referenced in the model space.
- An **advanced** detail - can be saved to the **Project** or in the **Library** and can be propagated in the model space.



1 Basic

2 Advanced

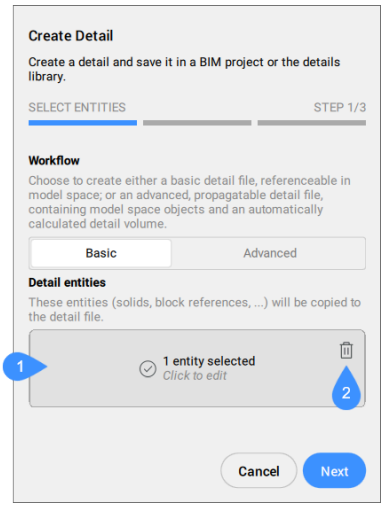
7.28.2 Basic

Creates a basic detail file with the selected entities in the BIM project, by simply specifying a detail name and a detail file name.

Note: Use the BIMADDDETAILREFS command to add references of the basic detail to the model.

The detail DWG file:

- Will be saved to the Project's files (the **Details** folder).
- Will be listed in the **BIM Project Browser** > **Files** tab under the **Details** folder, marked as a basic detail (right-click its name to access the options in the context menu).



1 Select in drawing

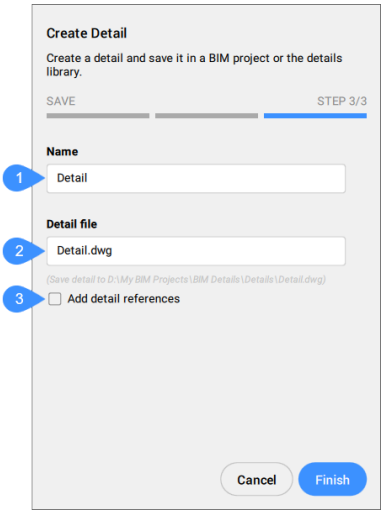
2 Clear selection

Select in drawing

Click this area to start the selection process. Press **Enter** to confirm the selection. Click again to edit the selection.

Clear selection

Click the recycle bin icon to clear the selection.



1 Name

2 Detail file

3 Add detail references

Name

Allows you to enter the name of the detail to be created. This name will be used when tagging the detail on sheets.

Detail file

Allows you to enter a detail DWG file name.

Add detail references
When checked, after saving the detail, BIMADDDETAILREFS is automatically launched, and the detail name will be passed via Command line argument to create a link between the master model and the detail file. So you can immediately start converting solids to volumes, or create volumes from (sub)entities extents.

7.28.3 Advanced

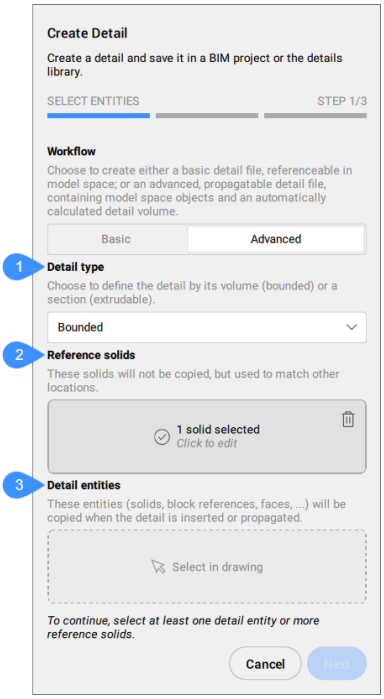
Creates a detail file that contains the necessary information (model space objects and an automatically calculated detail volume) to propagate the detail in the model.

Note: Use the BIMPROPAGATE command to propagate the advanced detail in the model.

The detail DWG file:

- Can be saved to the Project's files (the **Details** folder) or to the library (the **Details** panel).
- Will be listed in the **BIM Project Browser** > **Files** tab under the **Details** folder (if saved to the Project's files), marked as an advanced detail (right-click its name to access the options in the context menu).

STEP 1/3 allows you to select the detail type (**Bounded** or **Extrudable**), the **Reference solids** and the **Detail entities**.



1 Detail type

2 Reference solids

3 Detail entities

Detail type
Allows you to choose the type of detail to create.

Bounded
The detail will be applied to the selected solids to a length defined by the detail length.

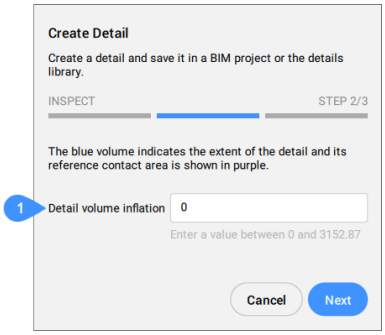
Extrudable
The detail will be applied to the whole length of the selected solids (for example, slab, wall, etc.)

Reference solids
Allows you to select the solids that define the situation where the detail should be applied. These solids will not be copied, but used to match other locations. At least one reference solid must be selected in order to continue to the next step.

Detail entities
You can optionally select entities to add more detail in relation to the selected reference solid (solids). These entities (solids, block references, faces...) will be copied when the detail is inserted or propagated.

STEP 2/3 allows you to set the **Detail volume inflation**.

Note: While on this step, set the optimal view for the detail in the model space. This view will be saved as the detail's thumbnail.



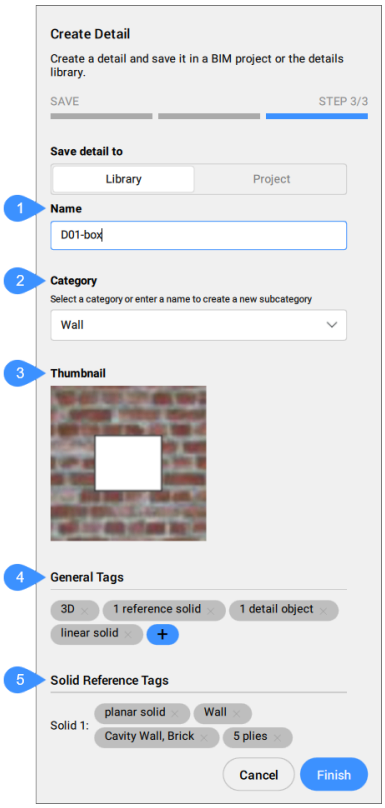
1 Detail volume inflation

Detail volume inflation
Expands the sensitive zone of the detail to include similar solids which are not connected yet in the model. The edges of the initially detected detail volume are moved outward by the specified factor.

STEP 3/3 allows you to select where to save the detail. The panel displays the settings corresponding to the selected location.

Save detail to
Allows you to choose where to save the detail.

Library
Saves the detail to the library. The detail can be accessed through the **Details** panel.



1 Name

2 Category

3 Thumbnail

4 General Tags

5 Solid Reference Tags

Name

Allows you to enter the name of the detail to be created. This name will be used when tagging the detail on sheets.

Category

Allows you to select a category or enter a name to create a subcategory.

Thumbnail

Displays the detail thumbnail created based on the current view.

General Tags

Displays the general tags, automatically and manually defined.

Add general tag

Lets you add custom general tags by clicking on the **Add tag** button. The custom general tags will differ in color from those automatically added.

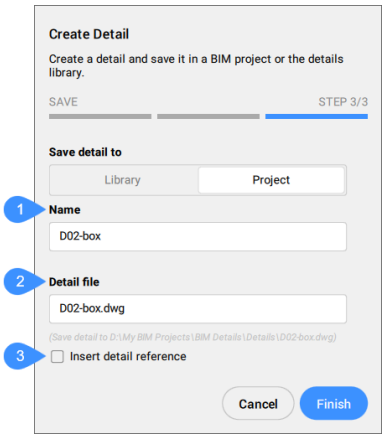
You can remove tags by clicking the corresponding **X** button.

Solid Reference Tags

Displays the automatically defined solid reference tags.

You can remove tags by clicking the corresponding **X** button.

Project
Saves the detail to the BIM Project's files. The detail can be found in the **Details** folder.



1 Name

2 Detail file

3 Insert detail reference

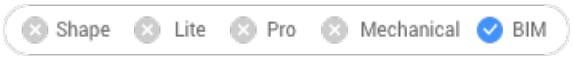
Name
Allows you to enter the name of the detail to be created. This name will be used when tagging the detail on sheets.

Detail file
Allows you to enter a detail DWG file name.

Insert detail reference
When checked, replaces the entities selected in the model for the detail creation with a detail reference.

7.29 BIMCURTAINWALL command

Creates a curtain wall.



Icon:



7.29.1 Description

Creates a curtain wall, as a block, from the face of a 3D Solid.

7.29.2 Method

Select a face and create a grid.

7.29.3 Options within the command

Length U panels
Sets the length of the panels in U direction (height).

Length V panels
Sets the length of the panels in V direction (width).

Number U panels
Number of panels in U direction (rows).

Number V panels
Number of panels in V direction (columns).

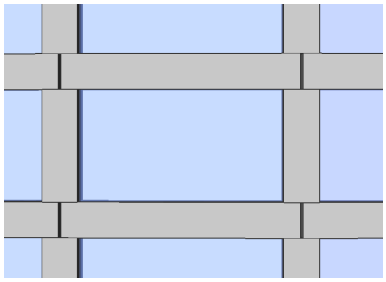
Width
Defines the width of the curtain wall frame, mullions and transoms.

Depth
Defines the depth of the curtain wall frame, mullions and transoms.

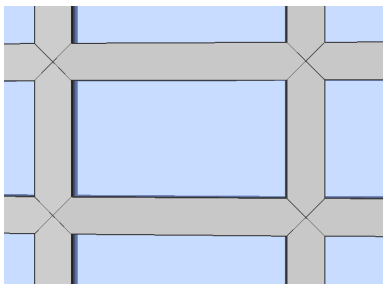
Glass thickness
Defines the thickness of the glass panels.

Connections type
Defines the type of the connections between horizontal and vertical mullions.

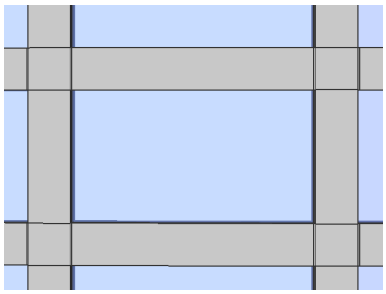
sTraight
Creates a straight connection.



Smooth
Creates a smooth connection.



Nodes
Creates a node connection.



Delete defining entities
Whether the selected source entity is retained or deleted depends on the value of the DELOBJ system variable.

7.30 BIMDECOMPOSE command

Decomposes compositions.

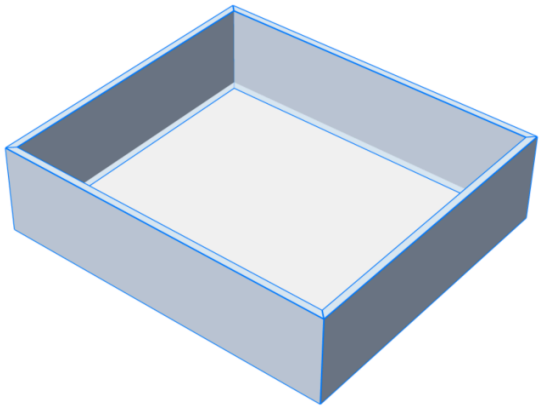
Shape Lite Pro Mechanical BIM

7.30.1 Description

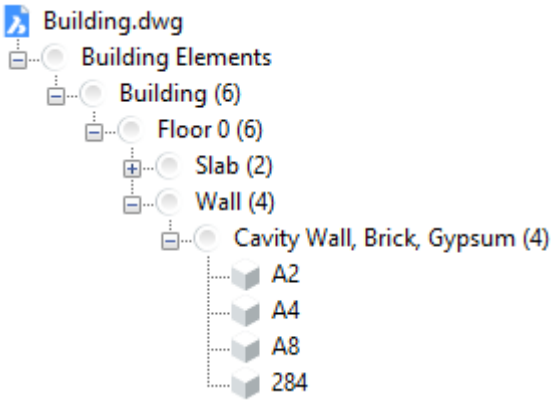
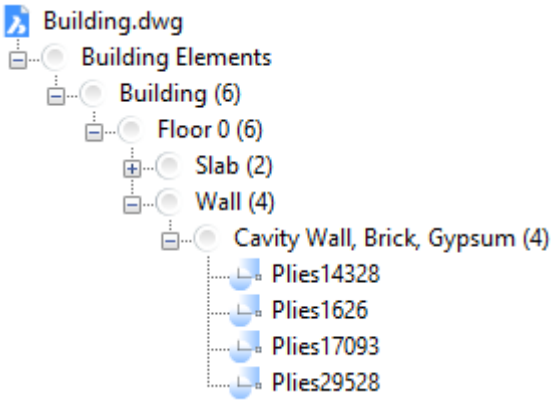
Decomposes compositions into their separate plies.

7.30.2 Method

This command offers one method to decompose compositions. After running the command, you can manually select BIM solids to decompose.



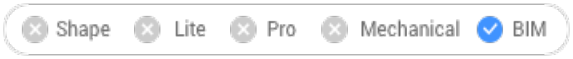
The solids are decomposed into their separate plies, which are put into one block. Each ply has inherited the BIM data of the original solid. You can see this in the **Structure** panel.

Before	After
	

Note: If you now want to manipulate the plies, you can use the BEDIT command.

7.31 BIMDIMENSION command

Dimensions the selected entities in the viewport semi-automatically.

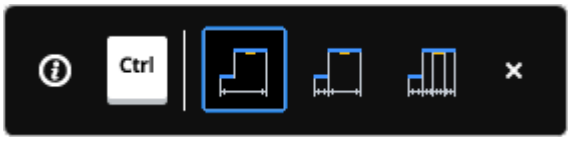


Icon:

Note: This command is only available in Paper Space.

7.31.1 Method

Select single or multiple entities in the viewport by clicking one by one or with Box selection, apply filters available in the **Dimension** dialog box and pick the location for dimensions. Press the CTRL key to toggle dimensioning options depending on the selected entities type.

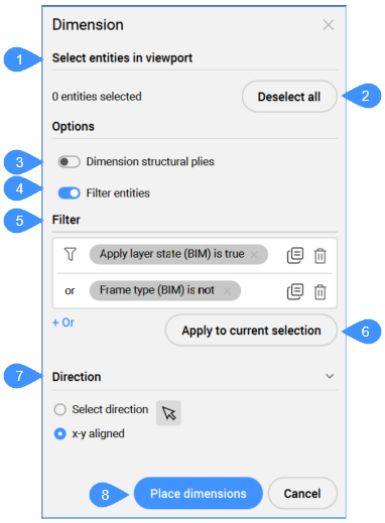


Note: Make sure that the Hot Key Assistant (**HKA** field in the **Status bar**) is ON.

Press Enter for a new selection set.

Note: You can also make a selection by using a python script to query specific objects (see the BIMPYTHON command).

This command opens the **Dimension** command panel.



1 Select entities in viewport

2 Deselect all

3 Dimension structural plies

4 Filter entities

5 Filter

6 Apply filter to current selection

7 Direction

8 Place dimensions

Select entities in viewport
Shows the number of selected entities.

Deselect All
Deselects all entities.

Dimension structural plies
Allows you to choose whether or not to dimension structural plies.

Note: Dimensions only plies that have the **Function** option set to **Structure**.

Filter entities
Toggles the display of the **Filter** selection section.

Filter
Allows to filter the selected entities by adding one or more parameter filters from the drop-down list. Filter rows can be added to filter by multiple parameter combinations. These rows are separated using the logical "OR" operation. The rows can be copied by clicking the copy symbol or deleted by clicking the delete symbol.

Apply filter to current selection
Applies the filter to the current selection.

Direction
Indicates the direction for the dimensions.

Select direction
Allows you to define a direction by pressing the arrow from the right side and specify an angle. The default direction is x, y axis.

x-y align
Dimensions will be aligned with x, y axis (horizontal or vertical).

Place dimensions
Specifies the place for dimensions.

7.31.2 Options within the command

place Dimensions
Pick a point to place the dimensions.

Undo
Undoes the last action.

deselect All
Removes the selection set.

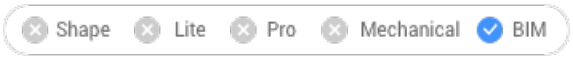
Select direction
Selects the direction for the dimensions.

x-y align
Dimensions will be aligned with x, y axis (default direction).

Angle
Dimensions will be oriented with the specified angle.

7.32 BIMDISTRIBUTIONSYSTEM command

Creates or edits a BIM Distribution system group.



7.32.1 Method

Enter an existing distribution system name or a name for a new distribution system, then select the entities to add to or remove from the distribution system.

Note: Distribution systems created with BIMDISTRIBUTIONSYSTEM command are exported with IFCEXPORT command as IfcDistributionSystem entities.

7.32.2 Options within the command

List distribution systems
Lists in the command line panel all existing BIM Distribution System groups.

Create a distribution system
Creates a new group of BIM Distribution System.

Show properties
Allows you to edit the properties. The available properties are displayed in the command line panel.

Ifc properties
Allows you to edit the IFC Properties. The available IFC Properties are displayed in the command line panel.

Delete distribution system
Deletes the selected BIM Distribution System group.

Create a distribution system
Creates a new BIM Distribution System group.

7.33 BIMEXTEND command

Extends a wall.

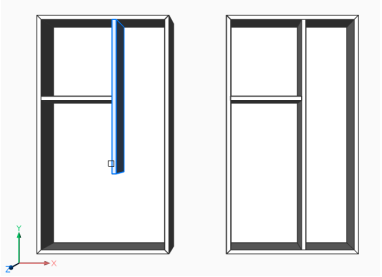
Shape Lite Pro Mechanical BIM

Icon:

7.33.1 Method

The command extends a side of a wall that doesn't intersect other walls by automatically detecting the nearest wall to extend it to.

Select 0-1 entities/subentities
Select the part of the wall that should be extended.



7.34 BIMFLIP command

Flips a starting face of a composition or mirrors/flips an insert.

Shape Lite Pro Mechanical BIM

Icon:

7.34.1 Description

Flips the starting face from which the layers of a composition are set out, or mirrors an insert (e.g. a window or door) left/right or flips it inside/outside.

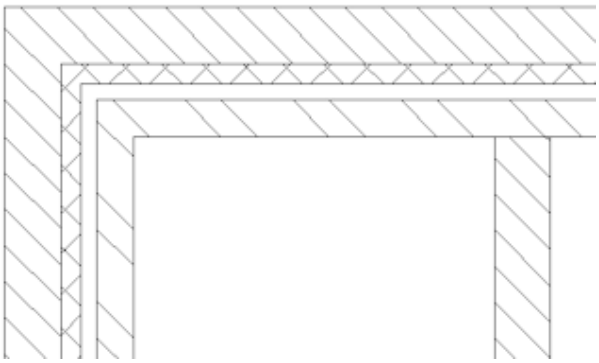
7.34.2 Method

There are two methods to flip entities:

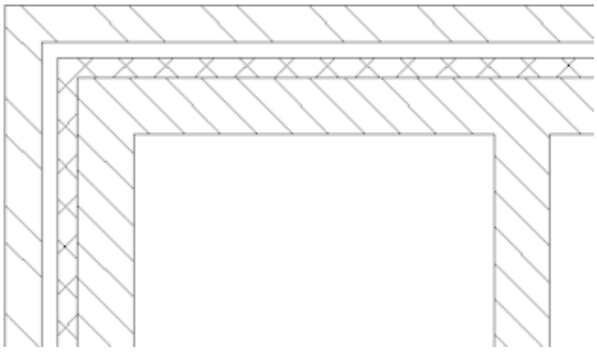
- Left- right.

- In-out

Before:



After:



Note: The result is visible in drawings generated by the BIMSECTIONUPDATE command if a multi-layered composition is attached to the sectioned solid.

7.34.3 Options within the command

Select BIM entity to flip
Inserts, walls and slabs are accepted in the command.

Note: The command only succeeds for solids that meet the following conditions:

- have a BIM composition attached.
- contain a BIM classification.
- are in a calculated BIM section.

Left-right
Mirrors the insert about a vertical axis in the face of the solid.

In-out
Flips the insert to the opposite face of the solid.

7.35 BIMFLOWCONNECT command

Creates a connection between flow segments.

Shape Lite Pro Mechanical BIM

Icon:

7.35.1 Description

Creates a connection between flow segments, such as pipes or HVAC ducts.

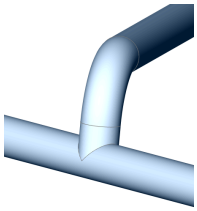
Note: When more than 2 flow segments are selected, the axes of the selected segments must be coplanar. When multiple coplanar segments are selected, the connections are created. When two non-coplanar solids are selected, an extra connection element is introduced.

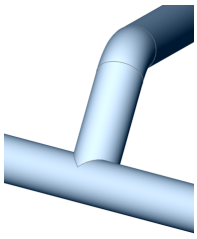
7.35.2 Options within the command

Switch

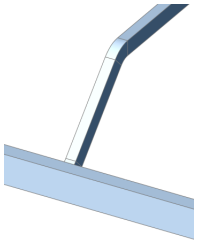
Switches between possible connections.

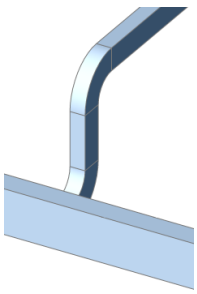
Alternatives for circular flow segments:

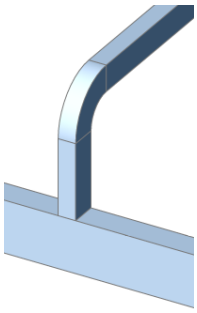




Alternatives for rectangular flow segments:

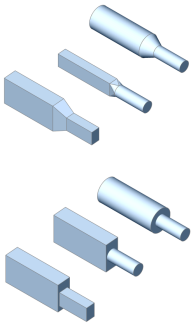






Note: Press the Ctrl key to cycle through different types of connection. Hot Key Assistant must be set ON.

When the section of the selected flows is different, reducer elements are inserted.
Alternatives for reducers:



Note: The creation of bends, tees and reducers is controlled by related system variables.

7.36 BIMGENERATE2DSTAIR command

Generates a symbolic 2D representation of a 3D stair by projecting the steps onto a plane.

Shape Lite Pro Mechanical BIM

Icon:

7.36.1 Description

Generates a 2D symbolic representation of a 3D entity classified as stair or as slab by projecting the steps or slabs to a plane.

Note: 3D entities classified as stairs can be:

- Stairs resulting from the BIMSTAIR command.
- Stairs which are created via direct modeling tools.
- Stairs imported from other software packages.
- Stairs created via Grasshopper scripts.

The automatically generated symbolic representation will be placed in the **BIM_2D_BACK+_Stair*** layers so that the solid geometry of the stair is still shown in the section result. Objects below the visible part of the stair will appear hidden. The layers can be customized and used during the section generation.

The direction of the stair is indicated in a 2d representation by an arrow pointing the up direction of the stair. The arrow starts at the first step and ends at the last step. A circle indicates the first step of the stair.

The numbering of steps starts with 1. The step text can be edited. Text style and layer settings can be set in the _SectionSettings.dwg file.

7.36.2 Options within the command

Select Section Plane
Allows you to select a Section Plane to project the stair or slab, to create a 2D stair or slab representation.

Continue without section plane
Creates a 2D stair or slab representation of the whole stair without selecting a Section Plane.

7.37 BIMGRID command

Creates 2-dimensional grids.

Shape Lite Pro Mechanical BIM

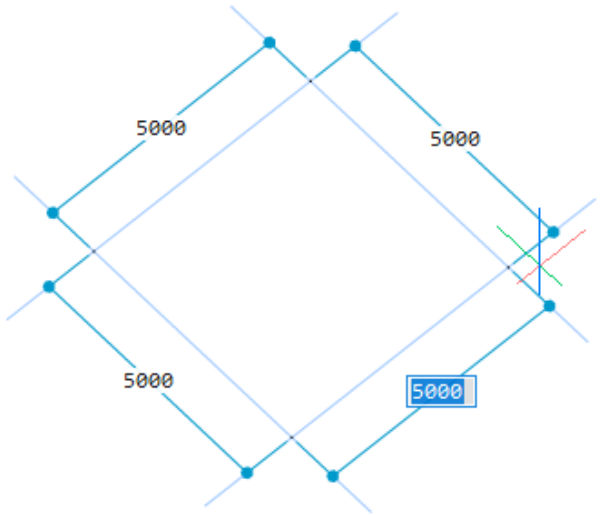
Icon:

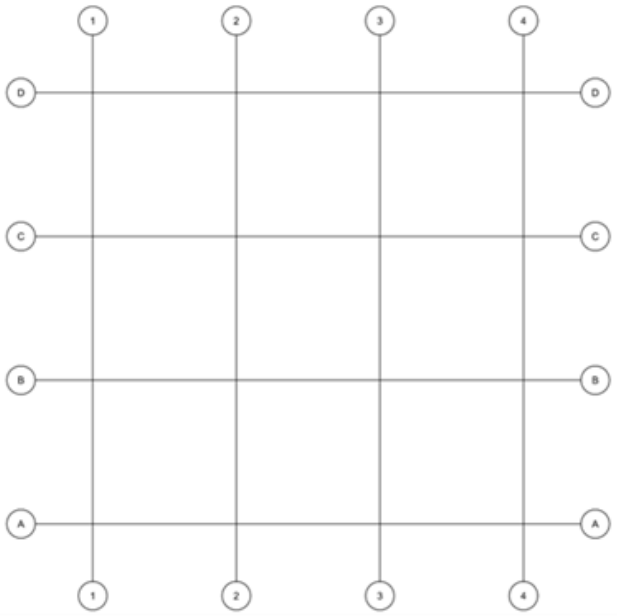
7.37.1 Description

Creates 2-dimensional rectangular and radial grids.

7.37.2 Method

The grid is created by specifying two points.





7.37.3 Options within the command

offsetU
Sets the line offset between U lines in V direction (interdistance between the U lines).

offsetV
Sets the line offset between V lines in U direction (interdistance between the V lines).

offset Labels
Sets the label offset to grid axes.

Radial
Creates a radial grid by specifying the center point, the start lines and the end lines.

7.38 BIMGROUP command

Creates or edits a Group with BIM elements.

Shape Lite Pro Mechanical BIM

7.38.1 Method

Enter an existing group name or a name for a new group, then select the entities to add to or remove from the group.

Note: Groups created with BIMGROUP command are exported with IFCEXPORT command as IfcGroup.

7.38.2 Options within the command

List groups
Lists all existing groups in the Command line.

Show properties
Allows you to edit the properties. The available properties are displayed in the Command line.

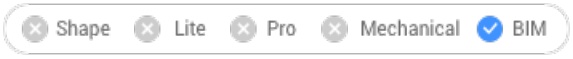
Ifc properties
Allows you to edit the IFC properties. The available IFC properties are displayed in the Command line.

Delete group
Deletes the selected group.

Create a group
Creates a new group.

7.39 BIMIFY command

Analyzes the model and runs automatic classification and spatial location assignment for the entire model.



Icon:

7.39.1 Description

Analyzes the model and runs an automatic classification and spatial location assignment for the entire model. Creates new buildings and/or stories and spaces if necessary. Optionally creates plan sections and elevations.

Note:

- Section entities are converted to BimSection entities.
- Simple block references can be classified.

7.39.2 Method

Running BIMIFY command on the entire model will classify objects, assign spatial locations, detect spaces and create elevations and floor plan section planes in the model.

When the BIMIFY process is completed, an overview of the result is provided in the Command line.

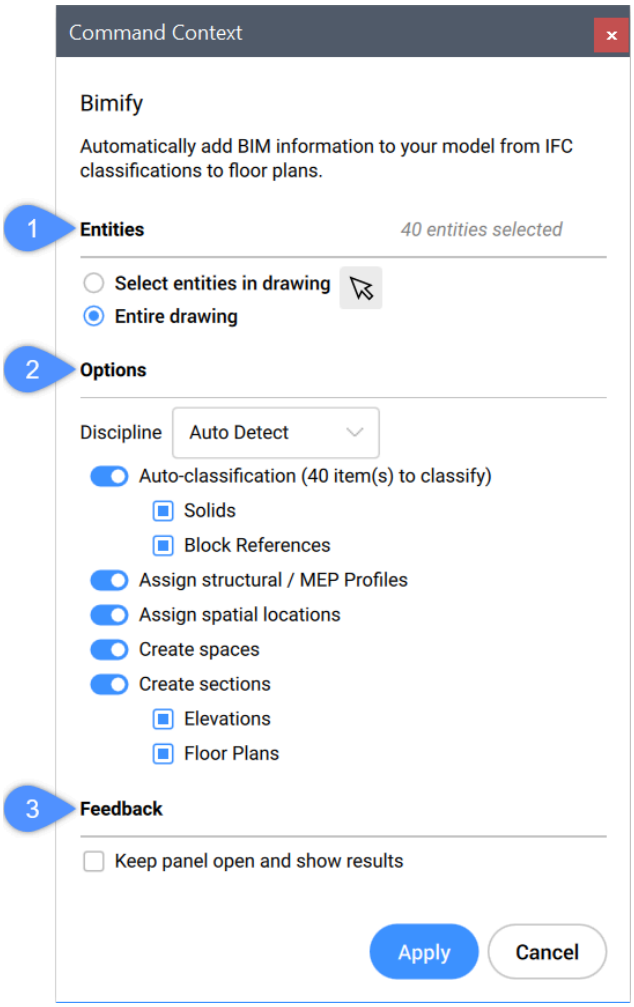
This command opens the **Bimify** command panel.

Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.
- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

7.39.3 Bimify command panel

The **Bimify** command panel allows you to analyze the model and run an automatic classification and a spatial location assignment for the entire model.



1 Entities

2 Options

3 Feedback

Entities

Select entities in drawing
The BIMIFY command runs for the selected entities in the drawing.

Entire drawing
The BIMIFY command runs for all entities in the drawing.

Options

Discipline
Determines to what discipline the model belongs 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.

Structural
The model is classified as a structural 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.

Auto-classification
Defines the classification of solids and blocks.

Note: Also see the BIMCLASSIFY command.

Solids
Makes sure that 3D solids are classified as walls, slabs, etc.

Block References
Makes sure that blocks are classified as windows, doors, etc.

Assign structural/MEP Profiles
Assigns columns, beams, members or flow segments profiles that are available in the **BIM Profiles** panel.

Note: 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.

Assign spatial locations
Assigns spatial locations (buildings and floors). See the BIMATTACHSPATIALLOCATION command.

Create spaces
Detects external walls and sets the property **Wall Common/is External** to **On** and also detects spaces. See the BIMSPACE command.

Create sections
Creates BIM Section entities. See the BIMSECTION command.

Elevations
Creates 4 elevations (Front, Back, Left and Right).

Floor Plans
Creates a plan section for each floor.

Feedback

Keep panel open and show results
If toggled **On**, the panel will show the progress and results while calculating.

Note: This option is **Off** by default for consistency with previous versions.

7.40 -BIMINSERT command

Inserts sheet metal form features, and mechanical and BIM components in the command line.

Shape Lite Pro Mechanical BIM

Note: This command is deprecated. Use -BMINSERT instead.

7.41 BIMINSERT command

Inserts BIM components into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command is deprecated. Use BMINSERT instead.

7.42 BIMINVERTSPACES command

Converts a set of input 3D solid entities and closed polylines, which represent spaces, to a building structure which encompasses these spaces.

Shape Lite Pro Mechanical BIM

Icon: 

7.42.1 Description

This command generates BIM elements in between a set of 3D solid entities and closed polylines. The result represents a building structure made of walls, openings, slabs, and roofs. You can define the options through the command context panel, as well as through the Command line.

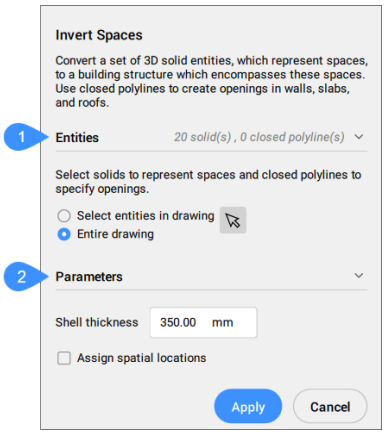
Note: This command is envisioned to be most useful in a Scan to BIM workflow. For more information, see the article **Point cloud Scan to BIM workflow**.

7.42.2 Method

The command takes closed polylines (created manually or using the POINTCLOUDFITPLANAR command) into account and creates parametric opening components based on these polylines. The polylines do not need to be coplanar with a face (exactly on the face) of the space. When polylines are detected on both sides of the wall, the two polylines are interpolated and the opening is created based on the interpolation results. The parametric opening object can then easily be replaced with a window or door component, using the BMREPLACE command.

Launch the command to open the **Invert Spaces** command context panel.

7.42.3 Options within the command context panel



1 Entities

2 Parameters

Entities
Displays the options for selecting the entities to be used as input.

Select entities in the drawing
Click on the arrow button to manually select 3D solids and closed polylines.

Entire drawing
All the 3D solids and closed polylines in the drawing are used as input.

Parameters
Displays the values of the parameters used to create the shell.

Shell thickness
Sets the thickness of the external walls created.

Assign spatial location
Tick the check-box to assign spatial locations to the created elements.

Note: The options within the command context panel reflect the options within the Command line.

7.43 BIMLINEARSOLID command

Creates a chain of linear solids.

Shape Lite Pro Mechanical BIM

Icon:

7.43.1 Options within the command

Set start point
Allows you to set a starting point for the linear solid. You can also use **Last point** and **Follow** as a starting point.

Angle
Draws at a user-specified angle and distance.

Length
Draws at a user-specified distance and angle.

Undo
Erases the last linear solid.

Quarter turn
Rotates the profile 90° counter clockwise.

Rotate
Allows to rotate the profile by a user defined angle.
Positive values rotate the profile counter clockwise.

choose other Profile
Allows to use a different profile for the next linear solid.

Last point
Press Enter to use the most recently specified point.

Follow
Draws the next linear solid at the same angle as the previous one.

Rectangular
Switches to a default rectangular profile.

Circular
Switches to a default circular profile.

Library
Opens the **Profiles** dialog box.

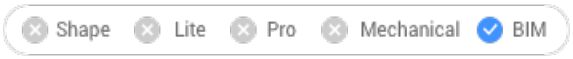
select in Model
Allows to select a profile in the model.

adJust profile
Allows to adjust the dimensions of the default rectangular and circular profiles.

Note: For the rectangular profile you can change the width and the height, and for the circular profile you can change the radius.

7.44 BIMLIST command

Lists the selected entities.



7.44.1 Description

Lists all the selected entities in the Command line, displaying their handle, material and GUID.

7.44.2 Method

Run the command and select the BIM entities to list.

7.45 BIMMULTISELECT command

Selects linear solids.

Shape Lite Pro Mechanical BIM

Icon:

7.45.1 Description

Selects linear solids with coplanar and/or parallel axes.

7.45.2 Method

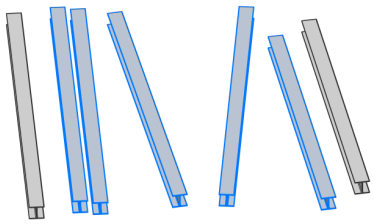
Selects linear solids with coplanar and/or parallel axes. If a face of a linear solid is selected, the corresponding faces of the linear solids with coplanar/parallel axes are selected.

Note: Select a different face by repeatedly pressing the TAB key to cycle through all faces of the selected solid.

7.45.3 Options within the command

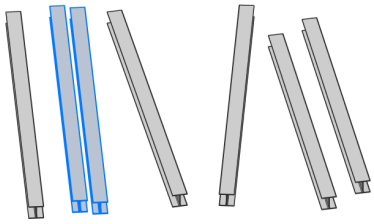
all Coplanar axes

Selects all solids of which the axis is coplanar with the axis of the selected solid.



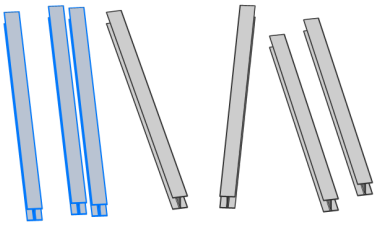
coplAnar and parallel axes

Selects all solids of which the axis is coplanar and parallel with the axis of the selected solid.



all Parallel axes

Selects all solids of which the axis is parallel with the axis of the selected solid.



Note: Press the Ctrl key to cycle through the command's options. Hot Key Assistant must be set ON.

7.46 BIMPARAMETRIZEDETAIL command

Generates parameters for a detail.

Shape Lite Pro Mechanical BIM

Icon:

7.46.1 Description

Generates parameters for a detail in order to easily change dimensions, angles, etc.

Note: Use this command in the detail file, not in a project. Detail files are saved in the folder, and its subfolders, which is defined by the DETAILSPATH system variable. By default, this is C:\ProgramData\Bricsys\Details\.

Note: The parameters and constraints can be edited in the Mechanical Browser.

7.47 BIMPROFILES command

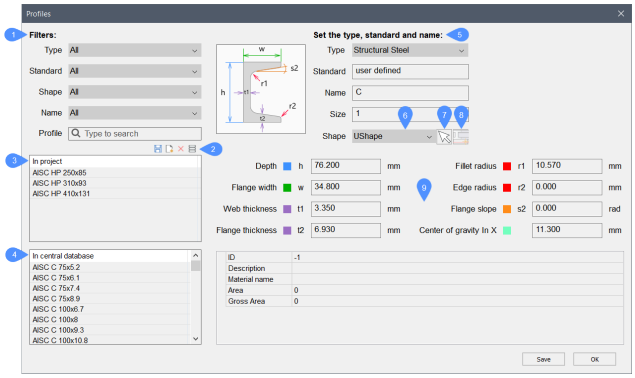
Opens the **Profiles** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

7.47.1 Description

Opens the **Profiles** dialog box to create and modify BIM profiles.



1 Filters

2 Tools

3 In Project

4 In central database

5 Custom fields

6 Preselect shape

7 Pick profile in model

8 Set profile offset

9 Properties of the profile

7.47.2 Filters

Allows you to filter the profiles list by type, standard, shape or name.

- **Type**: the options are **All**, **HVAC**, **Piping**, and **Structural Steel**. The **Electrical**, **Generic**, and **Structural Concrete** options become available for filtering if new profiles with these types are created.
- **Standard**: the Localized **Standard** options are **All**, **AISC** (American Institute of Steel Construction), **AS** (Australian Steel), **BS** (British Steel), **CNS** (Chinese National Standard), **EURO** (European standard steel profiles), **GOST** (Russian Steel Standard), **HVAC**, **JIS** (Japanese Industrial Standards), **Pipe**, **SAISC** (Southern African Institute of Steel Construction), and **STO ASChM** (Russian Steel Standard). The **user defined** option becomes available for filtering if new profiles are created.
- **Shape**: the options are **All**, **Circle**, **Circle Hollow**, **Custom Shape**, **I Shape**, **L Shape**, **Rectangle**, **Rectangle Hollow**, **T Shape** and **U Shape**. Depending on the selected standard, some shape options are not available. The **AsymmetricIShape**, **CraneRailAShape**, **CraneRailFShape**, **Ellipse**, **RoundedRectangle**, **Trapezium**, **ZShape** options become available for filtering if new profiles with these shapes are created.
- **Name**: filters the profiles by a selected character string.
- **Profile**: selects a character string. Only profiles that contain the selected string are listed.

7.47.3 Tools

- **Save Profile**: saves the currently edited profile.
- **New Profile**: creates a new profile.
- **Delete Profile**: deletes the selected profile.
- **Project and Library Information**: opens the **BIM Project Info** dialog box. Press the **Import** button to import profiles with XML/CSV files in project database (BSYSLIB).

7.47.4 In Project

Lists the available profiles used in the current project with respect to the above filters.

7.47.5 In Central Database

Lists the available profiles with respect to the above filters.

7.47.6 Custom fields

Lists the characteristics of the selected profile, or in case of creating a new profile, sets the type, standard and name for it.

7.47.7 Preselect shape

Selects a shape from the drop-down list.

7.47.8 Pick profile in model

Select a closed 2D entity or the boundary entities of an enclosed area.

7.47.9 Set profile offset

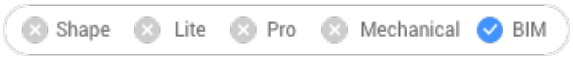
Sets the profile offset.

7.47.10 Properties of the profile

Shows the properties of the selected profile or sets the properties of a new profile.

7.48 BIMPROJECTINFO command

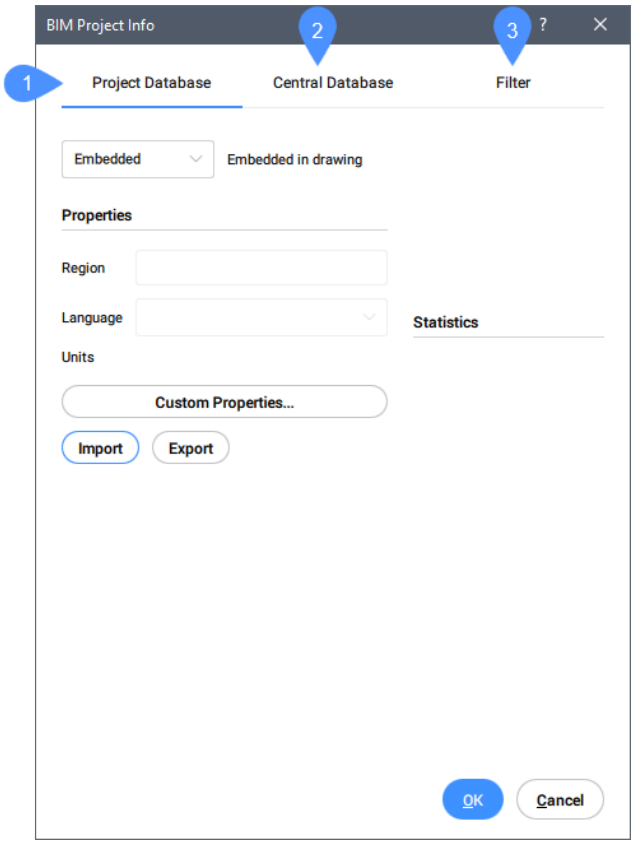
Opens the **BIM Project Info** dialog box.



7.48.1 Description

Opens the **BIM Project Info** dialog box to view and modify BIM project information.

The **BIM Project Info** dialog box shows information about properties, materials, compositions, profiles and spatial structures of the Project and Central databases. It allows you to import or export the database and set the project database as embedded or external. 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 central databases of BIM models can be changed using this dialog box.



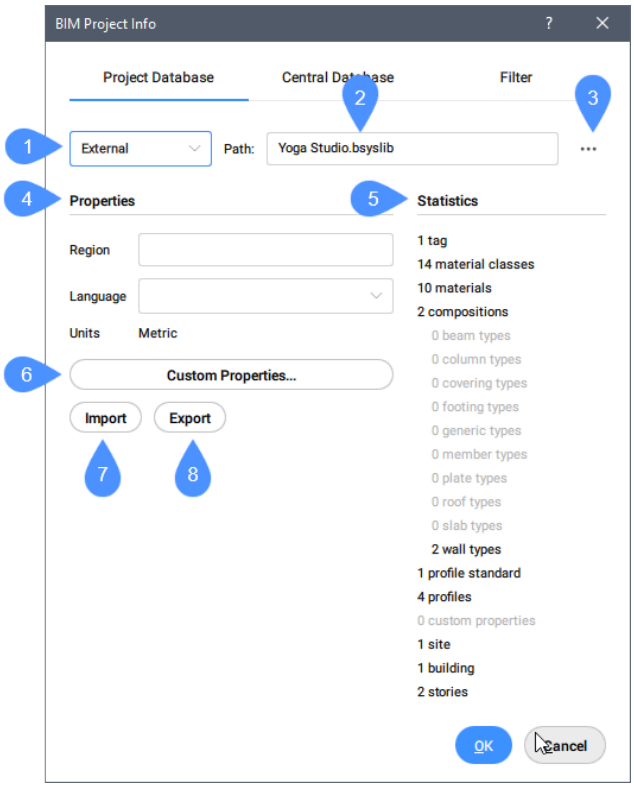
1 Project Database

2 Central Database

3 Filter

7.48.2 Project Database

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 database in the drawing file.

External
Saves the project database in a BIM database file (.bsyslib).

Note: The external database keeps the database independent from the current drawing. This is especially useful if you want to share the database between multiple models, e.g. when a model consists of several Xref drawings.

When switching from **External** to **Embedded**, the content of the external database is copied to the embedded project database.

Path
Displays the path of the project database.

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, spatial structures (the amount of buildings, stories, etc.) and the content of the current project database of the BIM model.

Custom properties…
Adds custom properties to materials and compositions. Opens the **Edit Custom Properties** dialog box (see **Edit Custom Properties dialog box** related article).

Import
Imports the selected *.xml or *.csv library files to enlarge the project database, for example, to add profiles. 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.

7.48.3 Central Database



1 Path

2 Browse

3 Properties

4 Set as default

5 Statistics

6 Custom Properties...

7 Import

8 Export

Path
Displays the path of the central project database.

Browse
Opens the **Select or create a Bricsys Library** dialog box.

Properties
Region, Language and Units are the properties of the project.

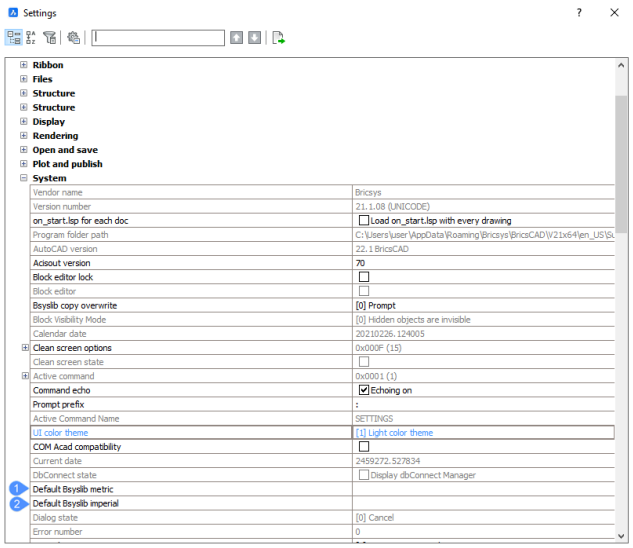
Region
Fill out the region field.

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.

Note: You can edit the default library database settings in the **Settings** dialog box under **Program options** > **System**.



Statistics
Shows the location, properties, spatial structures (the amount of buildings, stories, etc.), the content of the central project database of the BIM model.

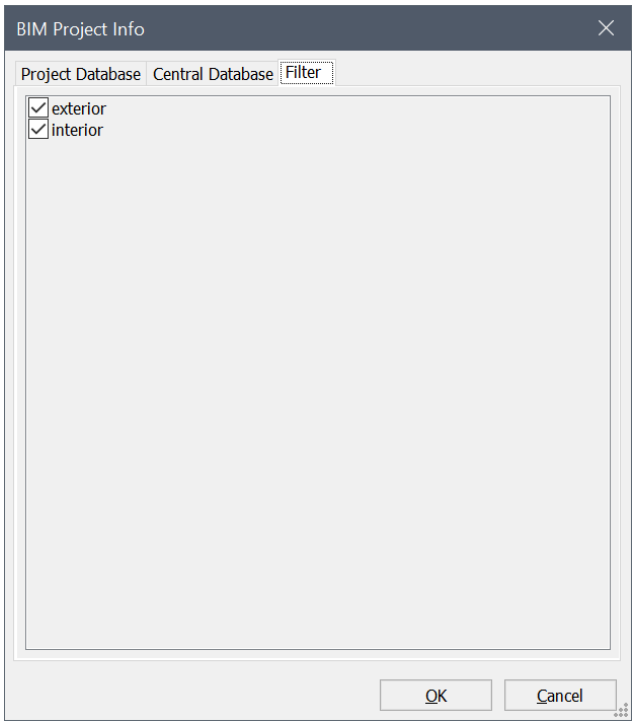
Custom Properties…
Adds custom properties to materials and compositions. Opens the **Edit Custom Properties** dialog box.

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.

7.48.4 Filter

Check the tags that you want to use in the filter.



7.49 BIMPROPAGATE command

Maps details related to selected base solids to similar base solids in the model.

Shape Lite Pro Mechanical BIM

Icon:

7.49.1 Description

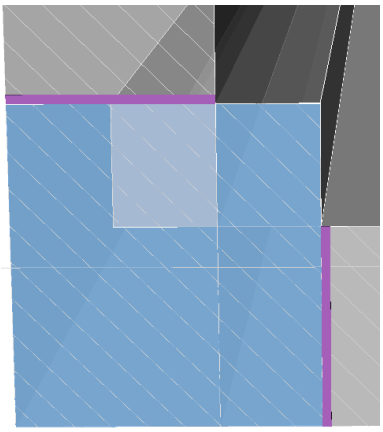
Maps details (solids, holes, finishing geometry, etc.) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid.

7.49.2 Method

Select reference solids and, optionally, detail entities (solids, block references, faces, edges, etc.).

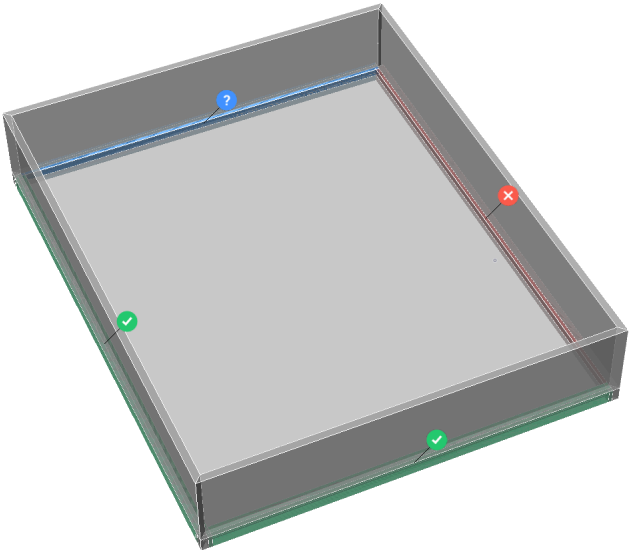
It can be used to copy details throughout your model.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick:

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



Note: Hovering over the widget will reveal more options.

Note: If the PROPAGATESEARCHSPACE system variable is ON, you can select a search space which allows you to limit the places that will be propagated.

Note: When an advanced project detail is propagated, the detail volume is used to add detail references at the places where BIMPROPAGATE modified/added detail geometry.

7.49.3 Options within the command

Block
Propagates the detail as a block.

Copy
Propagates the detail as a copy.

Select search space
Select the space(s) to limit the places that will be propagated.

Entire drawing
Selects the entire drawing as a search space.

No
Does not accept the detail.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the **Save Drawing As** dialog box, which allows you to save the detail.

7.50 BIMPROPAGATECORNER command

Propagates details.

Shape Lite Pro Mechanical BIM

Icon: 

7.50.1 Description

Propagates details connected to three planar base solids (e.g. corners).

7.50.2 Method

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar corners.

7.50.3 Options within the command

Select detail objects
Select additional solids, blocks or faces to be included in the corner detail.

yes, copy as Block
This option is available only when detail objects are selected. A block is created from the selected detail objects, which is then copied.

Note: The default block names are Block, Block 1, Block 2, ... You can rename blocks in the Blocks category in the Drawing explorer. See the EXPBLOCKS command.

yes, plain Copy
This option is available only when detail objects are selected. The detail objects are copied as is.

No
Exits the command without accepting the detail.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
The **Save Drawing As** dialog box displays.
You are prompted to specify a file name to save the detail.

Apply
Accept or reject suggestions by clicking on the tick.

Note: See the Apply option in the BIMPROPAGATE command.

7.51 BIMPROPAGATEEDGES command

Propagates a detail along the edge of a planar solid.

Shape Lite Pro Mechanical BIM

Icon:

7.51.1 Description

It propagates railings, gutters, borders, wall caps, etc.

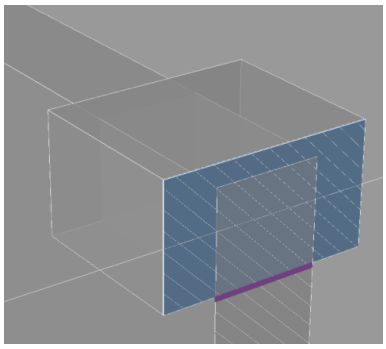
Note: The detail of BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

7.51.2 Method

Select planar reference solid to which the edge detail is related and the detail entities to be copied.

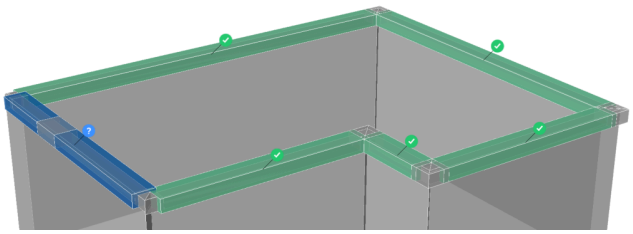
It can be used to copy details along edges of planar solids.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



7.51.3 Options within the command

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the **Save Drawing As** dialog box, which allows you to save the detail.

Yes
Only edges with similar orientation are selected.

No
All minor faces of similar planar solids are selected.

Outside reference solid
The propagated detail is placed outside the base solid. The volume of the base solids is not modified.

Inside reference solid
The propagated detail is placed inside the base solids. The volume of the detail is subtracted from the base solids.

7.52 BIMPROPAGATEFROMFILE command

Opens the **Select source file** dialog box.

Shape Lite Pro Mechanical BIM

7.52.1 Description

Opens the **Select source file** dialog box to select a DWG file to propagate in the current drawing.

After selecting the file and choosing **Open**, you are prompted at the Command line. The prompt sequence depends on the type of detail you selected: **Planar**, **Linear**, **Pattern**, **Edge** or **Corner**. For more information see the relevant BIMPROPAGATE commands.

7.53 -BIMPROPAGATEFROMFILE command

Propagates a saved detail.

Shape Lite Pro Mechanical BIM

7.53.1 Description

Propagates a saved detail on all suitable connections in your project.

Note: The prompt sequence depends on the selected detail type: planar, linear, pattern, edge or corner.

7.53.2 Method

Specify the full path and file name of the saved detail.

You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.

7.54 BIMPROPAGATELINEAR command

Propagates connections between linear elements.

Shape Lite Pro Mechanical BIM

Icon:

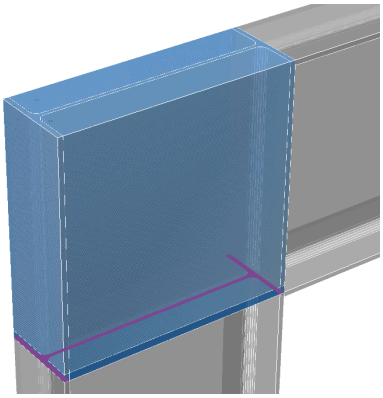
Linear elements accepted: beams, columns, pipes, ducts and their connections to walls and slabs.

7.54.1 Method

Select the linear or planar reference solids that form the connection and the detail entities to be copied.

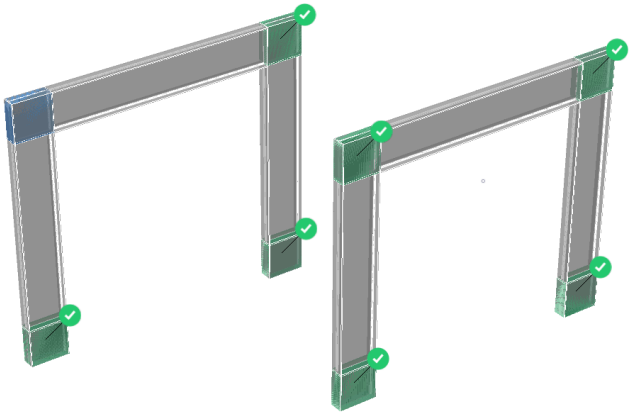
It can be used to copy detailed connections between two or more linear solids.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



7.54.2 Options within the command

Block
Propagates the detail as a block.

Copy
Propagates the detail as a copy.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the **Save Drawing As** dialog box, which allows you to save the detail.

7.55 BIMPROPAGATEPATTERN command

Propagates a single element on a flat surface to multiple locations and grids.

Shape Lite Pro Mechanical BIM

7.55.1 Description

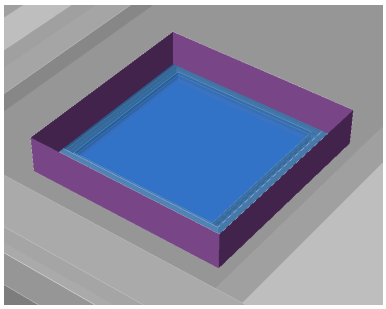
It propagates: lights, light switches, windows, air diffusers, etc.

7.55.2 Method

Select planar solid to which detail is related and the detail entities to be propagated.

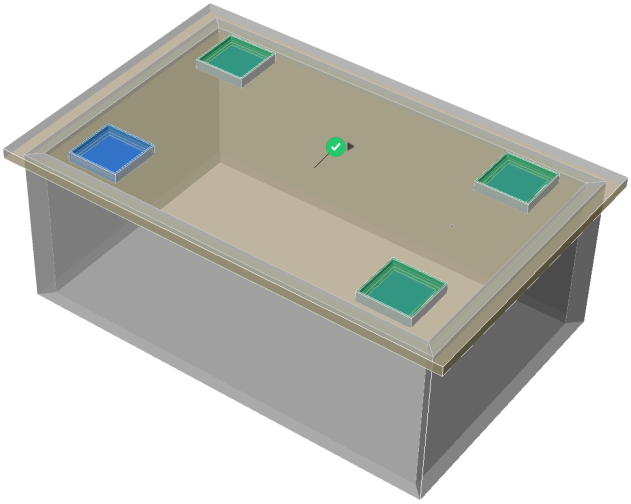
It can be used to copy objects to similar locations or over a certain pattern or grid on top of a planar solid.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



Note: Hover over the tick to toggle between Similar location and Grid options.

7.55.3 Options within the command

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the **Save Drawing As** dialog box, which allows you to save the detail.

7.56 BIMPROPAGATEPLANAR command

Propagates connections between planar elements.

Shape Lite Pro Mechanical BIM

Icon:

Planar elements accepted: walls, slabs, roofs.

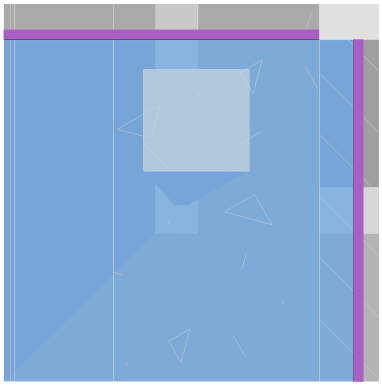
Note: The detail of BIMPROPAGATEPLANAR will always be a 2D slice through a linear detail.

7.56.1 Method

Select planar reference solids that form the connection and, optionally, detail entities to be copied as part of the connection.

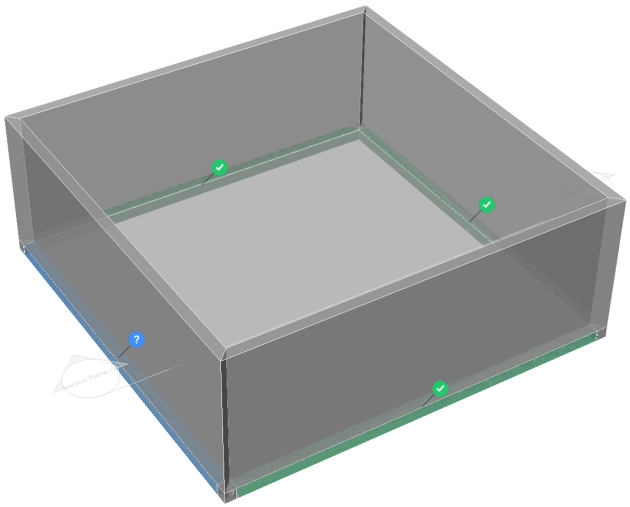
It can be used to copy detailed connections between two or more planar entities.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



7.56.2 Options within the command

Inflate first

Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail

Displays the **Save Drawing As** dialog box, which allows you to save the detail.

7.57 BIMPROPERTIES command

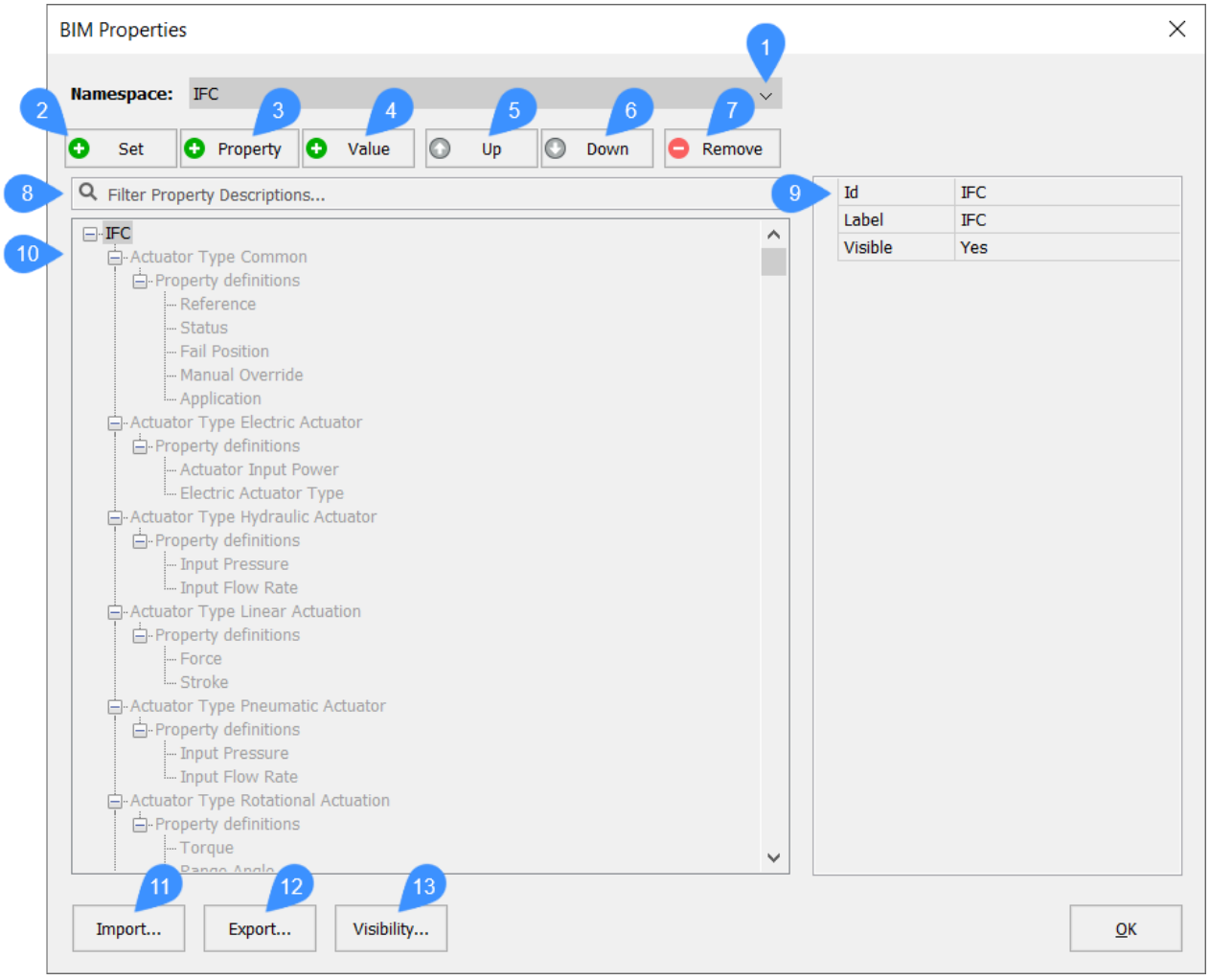
Opens the **BIM Properties** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

7.57.1 Description

Opens the **BIM Properties** dialog box 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 Up

6 Down

7 Remove

8 Filter

9 Properties

10 Properties Tree

11 Import

12 Export

13 Visibility

7.57.2 Namespace

By default, the below namespaces are available:

- **User**: allows you to create user defined properties.
- **IFC**: shows the IFC2x3 and IFC4 properties merged under a single namespace.

  Note:

  - Differences between the IFC schemas will be dealt with in the background during IFC Import or Export.
  - Static attributes from the BIM namespace are marked as deprecated not to be used instead of dynamic ones.
- **Quantity**: shows the quantities properties. It is read-only.

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.

7.57.3 Add Set

Creates a new property set.

7.57.4 Add Property

Adds a new property to the currently selected property set.

7.57.5 Add Value

Creates a new value definition.

7.57.6 Up

Moves up the selected property or value.

7.57.7 Down

Moves down the selected property or value.

7.57.8 Remove

Removes the selected property set, property definition value definition, or namespace.

Note: In case of removing a namespace, a warning message displays.

7.57.9 Filter

Displays only the properties of which the Id and Label contain the search string typed. The results in the properties tree are filtered as you type.

7.57.10 Properties

Displays the properties of the selected property sets, property definitions and value definitions.

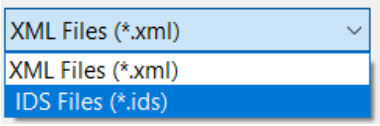
Note: Visibility rules for the property definitions are available. It is possible to make the visibility of a property or property set dependent on the value of another property.

7.57.11 Properties Tree

Displays the property sets, property definitions and value definitions.

7.57.12 Import

Imports property definitions based on the XML or IDS-XML schema definitions. Opens the **Select XML file to import** dialog box, where you can choose the file to import, be it XML or IDS.



An XSD (XML Schema Definition) file has been created to generate XML files for BIM properties. The file can be accessed here.

The IDS-XML property definition contains IFC properties (defined by buildingSMART®), custom properties, and IFC attributes. An IDS namespace is created. When there are properties in the IDS file that are not part of the IFC Pset properties, they will be added as new property definition to the IDS namespace.

Note:

- When an IDS file is imported, the SHOWIDSPROPERTIESONLY system variable is automatically set to ON. This toggles on the **IDS Properties** option in the **Visibility** dialog box.
- When the SHOWIDSPROPERTIESONLY system variable is ON, only the custom properties, added to the IDS namespace, as well as the IFC properties in the IFC namespace that were part of the IDS file are displayed in the **Properties** panel (hiding all IFC properties that are not specified in the imported IDS file).

7.57.13 Export

Exports to an XML file only the active namespace (selected in the drop-down menu) with its property sets (properties/quantities) as specified by the IFCEXPORTIDSPROPERTIESONLY system variable:

- Export all active namespace's properties (OFF).

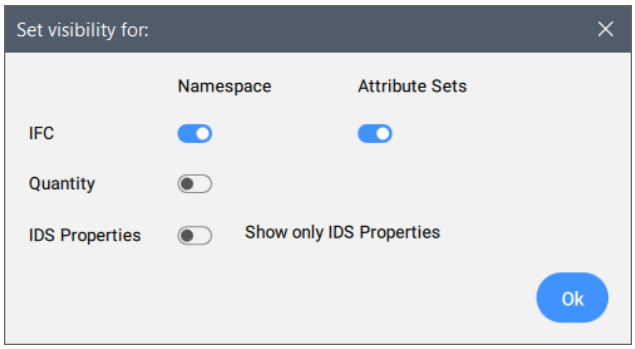
- Export only the active namespace's IFC and custom properties mentioned in the imported IDS file (ON).

This file can then be imported in other drawings.

Note: If the IFCEXPORTMULTIPLYELEMENTSASAGGREGATED system variable is set to ON, value definitions for visible Ply-elements are also exported.

7.57.14 Visibility

Enables you to toggle on and off the visibility of IFC **Namespace** and **Attribute Sets**, **Quantity** and **IDS Properties**.



Note: The **IDS Properties** toggle changes the value of the SHOWIDSPROPERTIESONLY system variable.

7.58 BIMPYTHON command

Allows Python Scripting and Querying of a BIM model.

Shape Lite Pro Mechanical BIM

Icon:

7.58.1 Method

Opens the **Select Python script** dialog box and runs the chosen script.

Note: Python version shipped with BricsCAD is increased to 3.9.6.

7.59 -BIMPYTHON command

Allows Python Scripting and Querying of a BIM model via Command line.

Shape Lite Pro Mechanical BIM

7.59.1 Method

Opens the **Select Python script** dialog box and runs the chosen script.

Note: Python version shipped with BricsCAD is increased to 3.9.6.

7.60 BIMQUICKBUILDING command

Transforms solids into buildings.

Shape | Lite | Pro | Mechanical | BIM

7.60.1 Description

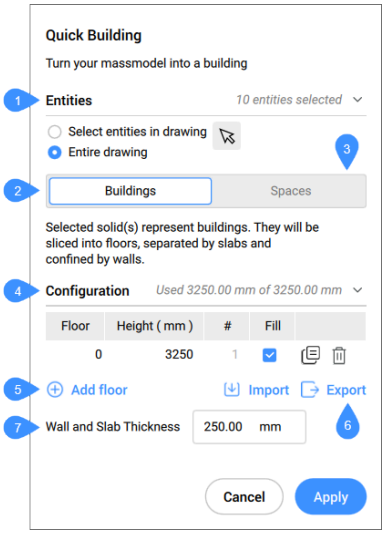
Creates a separate drawing for the generated BIM building. In this drawing, walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created.

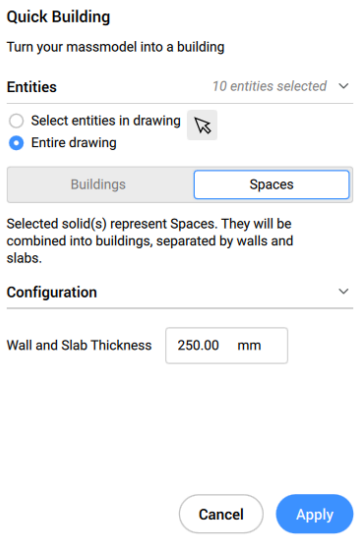
Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.
- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

7.60.2 Method

The command opens the **Quick Building** command context panel.





1 Entities (selected)

2 Building mode

3 Spaces mode

4 Configuration

5 Add floor

6 Import / Export

7 Wall and Slab Thickness

Entities (selected)

Select entities in the drawing
Allows you to choose which entities to select.

Entire drawing
By default, the panel selects all 3D solids in the model space.

Buildings mode
Overlapping input solids will be unionized and for each resulting solid a building will be created. Additional input is needed to divide the building in multiple storeys. They will be sliced into floors, separated by slabs and confined by walls.

Configuration
Displays the elevation table.

- **Floor**: displays the storey number.
- **Height**: sets and displays the storey height (elevation value).
- **#**: sets and displays the number of equal storeys height (number of floors).
- **Fill**: if checked, an automatic number of storeys are generated according to the elevation input (**Height**) value and the remaining available height (elevation value).
- : duplicates a storey.

- : deletes a storey.

Add floor
Adds a storey.

Import / Export
Imports from or exports to an elevation table as CSV format file.

Wall and Slab Thickness
Sets and displays the value of the wall and slab thickness.

Note: The elevation input (**Height**) and **Wall and Slab Thickness** respects the insertion units of the drawing.

Spaces mode
The individual input solids will be considered as spaces in a building. For each group of touching solids, a building will be created. Each resulting building will consist of floor slabs and inner walls, corresponding to the touching face pairs of the input solids.

7.60.3 Options within the command

change Selection
Allows you to select entities in the drawing, as, by default, the entire drawing is selected.

Import
Uses a CSV or TXT file to configure the storey heights of the building.

Note: For instance, a text line for 2 storeys configuration can be:

```
Floor;Height- mm;Amount;Fill
 0 - 10;3250.000000;11;Yes
 12 - 16;5000.000000;5;No
```

sPaces
Generates spaces for buildings.

7.61 BIMQUICKDRAW command

Creates and edits spaces and buildings.

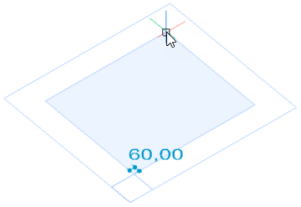
Shape Lite Pro Mechanical BIM

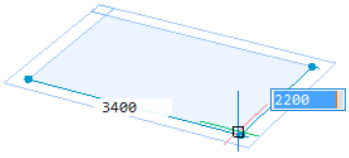
Icon:

7.61.1 Description

The command quickly creates and edits rectangular and L-shaped spaces and buildings. The layout of the **QuickDraw** cursor represents the space being created. It also indicates the alignment with existing walls and distances from existing solids.

Note: If dynamic input (**DYN**) is set to **On**, the dimensions are displayed as the spaces are created. The dimensions can also be entered manually.



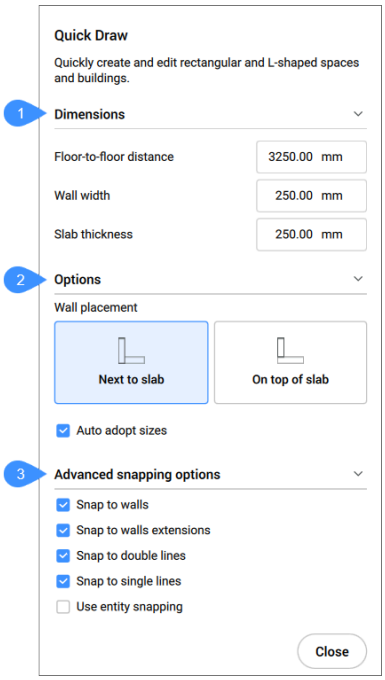


Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.
- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

7.61.2 Method

The command opens the **Quick Draw** command context panel. The **Hotkey Assistant** widget appears as well.



1 Dimensions

2 Options

3 Advanced snapping options

The **Hotkey Assistant** widget lets you change the justification of the walls.



7.61.3 Dimensions

Floor-to-floor distance (Floor height)
Sets the distance between the elevations of two subsequent floors.

Note: The default value is 3250mm.

Wall width
Sets the width of the walls being created.

Note: The default value is 250mm.

Slab thickness
Sets the slab thickness of the room.

Note: The default value is 250mm.

Note: The dimensions set in the command context panel become the new default values.

7.61.4 Options

Wall placement
Allows you to set the connection type between the slab and the outer walls.

- **Next to slab**: places the wall next to the slab.
- **On top of slab**: places the wall on top of the slab.

Auto adopt sizes
Controls whether wall width, wall height and slab thickness should be adopted from nearby rooms.

- **ON**: the width and height are adopted from the highlighted wall when you place the QuickDraw cursor against a wall to specify the first corner of a new room. When two walls of different height or width highlight (corner), the dimensions of the wall that highlights first are adopted.
- **OFF**: the dimensions used are those specified in the command panel.

7.61.5 Advanced snapping options

Snap to walls
Controls whether the room cursor should snap to walls.

Snap to walls extensions
Controls whether the room cursor should snap to extensions of walls.

Snap to double lines
Controls whether the room cursor should snap to parallel double lines placed a 'wall-width' apart.

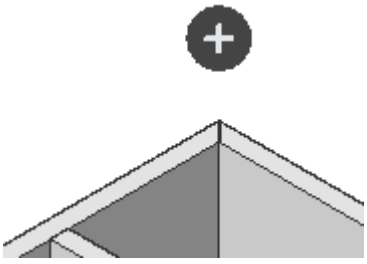
Snap to single lines
Controls whether the room cursor should snap to single lines (for example grid lines, etc.).

Use entity snapping
Controls whether the room cursor should also snap to entities using **ESNAP**.

Cancel
Closes the command panel and ends the command.

7.61.6 Add story widget

You can use the widget to add a story to an existing building. For each new story added, you are able to select the desired wall placement configuration from the command context panel (or use the current setting), before choosing one of the options:



-
  Copies the top floor entirely.

- 

  Copies the slab and the outer walls of the top floor.
- 

  Finishes the building with a flat roof.

7.62 BIMREASSOCIATE command

Automatically re-associates invalid Tags and Dimensions to the underlying geometry.

7.62.1 Description

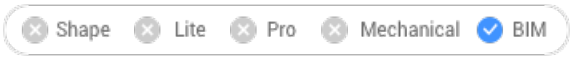
Select a BIM section viewport in order to automatically re-assign invalid Tags and Dimensions to the underlying geometry.

Note: The BIM **Associativity** property indicates whether a tag is associated.

Note: The BIM **Associative coloring** property visualizes an unassociated tag in red, if true. It doesn't change the color property. After re-associating, BIM tags are displayed in their layer color.

7.63 BIMRECALCULATEAXIS command

Recalculates and repositions the axis of linear building elements.



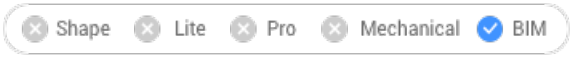
Icon: 

7.63.1 Description

Recalculates the axis of linear building elements and repositions it back on the center line of a linear element. When the command is run, select all linear solids for which to recalculate the axis.

7.64 BIMROOF command

Creates solids classified as **Roof**.



Icon: 

7.64.1 Description

The command creates a roof based on a closed 2D curve or boundary. You can define the options through the command context panel, as well as through the Command line.

7.64.2 Method

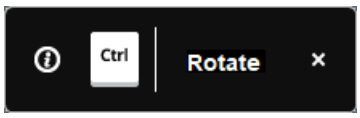
Launch the command to open the **Roof** command context panel.

There are three methods to define the out limits of a roof:

- Draw a rectangle.

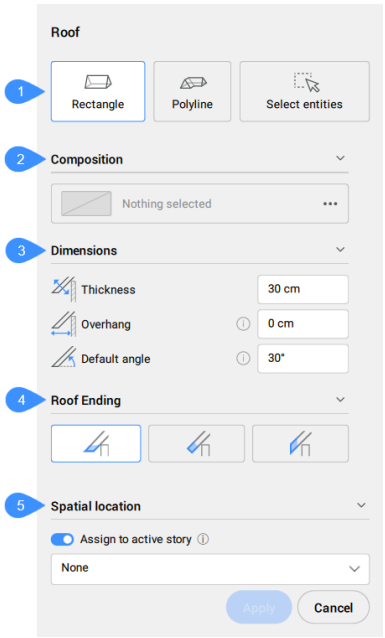
- Draw a polyline.
- Select entities in the drawing.

Use the **Hotkey Assistant** widget to change the direction of the current roof. Press the **Ctrl** key to rotate the direction by 90 degrees CCW.



Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMROOF options** check box is ticked in the **Hotkey Assistant Configuration** dialog box (see **Hotkey Assistant widget** article).

7.64.3 Options within the command context panel



1 Creation mode

2 Composition

3 Dimensions

4 Roof ending

5 Spatial location

Creation mode
Allows you to choose a method for creating the roof.

Rectangular
Define the new roof's out limits by manually drawing a rectangle.

Polyline
Define the new roof's out limits by manually drawing a polyline. Press the **Enter** key to close the polyline.

Note: Arc segments are not supported as input.

Select entities
Define the new roof's out limits by selecting entities in the drawing. Select either a single 2D curve/region/boundary or multiple lines/walls, then press **Enter**. When entities are preselected before launching the BIMROOF command, the panel automatically selects this mode.

Composition

Click the browse button (...) to open the **Compositions** dialog box that lets you define the current roof composition. By default the compositions of type **Roof** are shown. You can change this filter in the **Compositions** dialog box.

Dimensions

Thickness
Sets the roof thickness.

Note: When a composition with a fixed or minimal thickness is selected, the thickness input field is constrained accordingly.

Overhang
Sets the horizontal distance between the outer roof edge and the wall.

Default angle
Sets the angle to generate the initial roof. Once the initial roof is created the angle for each roof plate is accessible in the dynamic input fields that appear in the model space.

Note: You can further set the angle for each roof plate by typing a new value in these dynamic input fields. Type in 0 or leave a field empty to indicate that no roof plate should be created in that particular area. Type in 90 to have the adjacent plates form a gabled roof.

Roof ending
Sets the type of roof ending: horizontal, perpendicular, vertical.

Spatial location
Allows you to select a spatial location from the drop-down menu to assign to the roof.

Assign to active story
Allows you to copy the spatial location of the active story which is currently active in **Top View Mode** (TVM) or of the closest story level if not in TVM.

Note: The options within the command context panel and those within the **Hotkey Assistant** widget reflect the options within the Command line.

7.65 BIMSECTION command

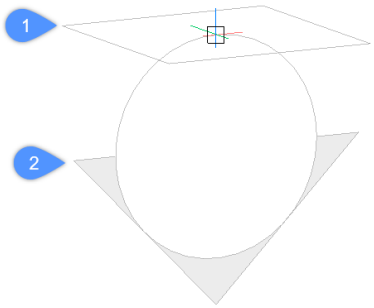
Creates a BIM Section entity.

Shape Lite Pro Mechanical BIM

Icon: 

7.65.1 Description

Creates a BIM Section entity with the section plane (1) in the XY-plane of the current coordinate system (WCS or UCS) and the view direction (2) in the negative Z-direction of the current coordinate system.



Note: If dynamic UCS (**UCSDETECT**) is **On**, the section plane aligns to the face of a 3D solid under the cursor.

BIM Section entities are created on the current layer. A separate BIM_SECTION layer is generated to store section indicators. For more information, see the Guide article Generating drawings.

7.65.2 Method

There are seven types of BIM Section entities that can be created using the BIMSECTION command:

- **Plan**: displays a horizontal section plane that cuts though the model.
- **Section**: displays a vertical section that cuts though the model.
- **Elevation**: displays the exterior elevation view of the model.
- **Detail**: displays a manually defined section volume.
- **Reflected Ceiling Plan**: displays a horizontal section plane that cuts though the model, with the view direction oriented downwards and the ceiling lines projected on this plane.
- **Interior Elevation**: displays an interior elevation view for each wall of the selected space.
- **Interior Floor Plan**: displays a horizontal section plane that cuts though the selected space, containing the associated interior elevations indicators.

Note: You can change the **Section type** property of the selected BIM Section entity from the **Properties** panel.

Note: The GENERATEASSOCVIEWS system variable controls if the VIEWSECTION command produces 2D drawings which are persistently associated with the source 3D model. If GENERATEASSOCVIEWS is **On**, associative dimensions are updated automatically when the 3D model is modified and BIMSECTIONUPDATE is executed.

7.65.3 Options within the command

Select a point to place section
Allows you to specify a point.

Note: The section plane displays dynamically parallel to UCS XY-plane, through the cursor position.

Specify distance
Allows you to type a distance or specify a point.

Note: It is recommended to have **Dynamic input** (DYN) active. It allows to enter the distance in the dynamic entry field.

Note: The section plane is defined through the point or at the specified offset from the first point. The section line is parallel to the X-axis of the UCS or dynamic UCS and through the point at the specified offset from the first point.

Note: Only the section line and the callouts of a BIM Section entity display. When highlighted or selected, the section plane, section boundary and/or section volume display as well.

set clipping oN
Sets the **Clip display** property to **On**.

Note: This option is set by default. If the **Hot Key Assistant** (HKA) system variable is **On**, press the **Ctrl** key to toggle the **Clip display** property **Off**.

Note: Multiple sections can have the **Clip display** property set to **On** simultaneously. This property can be saved in a model view (see the VIEW command).

set clipping Off
Sets the **Clip display** property to **Off**.

Note: You can change the **Clip display** property in the **Properties** panel.

Detail
Creates a **Detail** section type. For more information, see the Guide article **Define detailed section**.

Based on an existing section
Select an existing section to position the base of your Detailed Section on. The base rectangle of the detail section will be parallel to the section plane of the selected section.

Interior
Creates interior elevations and a floor plan of the selected spaces.

Note: The **BIM/Interior elevation** property of a space is set to **On**. Use the BIMSECTIONUPDATE command to update the interior elevation.

Attach section
Select existing sections to attach as Interior Elevation.

Scale
Sets the **Scale** property of the viewport in the drawing file created by the BIMSECTIONUPDATE command.

Note: The default **Scale** is saved through the SECTIONSCALE user preference (default value is 0.02) in the **Settings** dialog box.

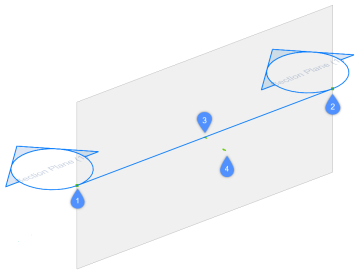
Reflected ceiling
Creates the plan of a ceiling projected onto the section plane.

Note: A reflected ceiling plan shows the size and location of light fixtures and other structures on the ceiling. The clip display property of a reflected ceiling section is **Off** by default.

7.65.4 Grips editing

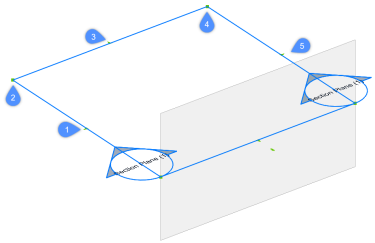
BIM Section entities can be edited through grips, depending on their **State** property. The **State** property can be changed from the **Properties** panel.

- **Plane** state:



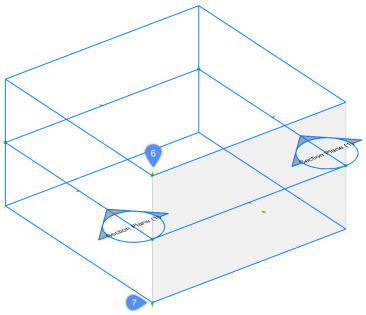
  - Start grip (1) allows you to move the section and the position of the Start callout.
  - End grip (2) allows you to modify the orientation of the section and the position of the End callout.
  - Center grip (3) allows you to move the section and the position of the Mid callout.
  - The arrow (4) flips the view direction.

- **Boundary** state:



  - Midpoint grip (1) allows you to move the section entity.
  - Corner grip (2 & 4) allows you to deform the section boundary/volume.
  - Midpoint grip (3 & 5) allows you to stretch the section boundary/volume.

- **Volume** state:



  - (6) allows you to move the top plane of the section entity.
  - (7) allows you to move the bottom plane of the section entity.

7.66 BIMSECTIONOPEN command

Opens the drawing file related to a BIM section entity.

Shape Lite Pro Mechanical BIM

Icon: 

7.66.1 Description

Opens the drawing file related to a BIM Section entity, or the 3D BIM model related to a BIM section drawing.

7.66.2 Method

There are two methods to use BIMSECTIONOPEN command:

- Select a BIM Section entity in Model Space to open the generated BIM section drawing.
- Select a BIM Section result (a viewport) in Paper Space to open the corresponding 3D BIM model.

7.67 BIMSECTIONUPDATE command

Updates the result of a BIM section entity.

Shape Lite Pro Mechanical BIM

Icon: 

7.67.1 Description

Generates or updates the drawing file related to a BIM Section entity.

Note: The GENERATEASSOCATTRS system variable controls the 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 to produce drawings for which dimensions and tags can be automatically updated when the 3D model is modified.

Note: When GENERATEASSOCVIEWS system variable is ON, the BIMSECTIONUPDATE command will generate/update the section indicators, grid callouts and story callouts.

Tags are not automatically generated with the BIMSECTIONUPDATE command.

Story indicators, section indicators and grid callouts are generated in paper space and are essentially BIM tags.

7.67.2 Method

There are two methods to use the BIMSECTIONUPDATE command:

- Generate the result of a BIM Section entity.

  Note: If the **Interior** option of the BIMSECTION command was used during the creation of the BIM Section entity, BIM Spaces can also be selected.

- Update the result of a BIM Section entity.

Note: Select a BIM Section entity in the Model Space or a viewport in the Paper Space layout.

After BIMSECTIONUPDATE, all BIM tags and annotations will have the new properties **Associativity** and **Associative Coloring** attached. When **Associativity** is **Not Associated** and **Associative Coloring** is **On**, the annotation will be visualized in red, but the native **Color** property of the entity will not be changed.

7.68 BIMSETLOADBEARINGDIRECTION command

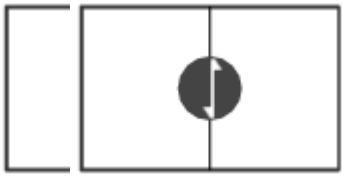
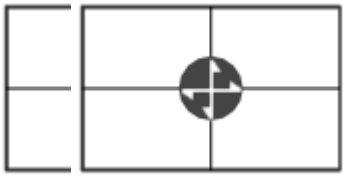
Sets the loadbearing direction for BIM slab solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.68.1 Method

An icon is displayed in the center of the slab after selecting a slab solid. Set the loadbearing direction for BIM slab solids by clicking on the icon.

None	Short	Long	Both
			

7.69 BIMSUBGROUP command

Adds or removes nested Groups, Building Systems, Distribution Systems or Zones.

Shape Lite Pro Mechanical BIM

7.69.1 Method

Enter the name of an existing group, building system, distribution system, or zone, or create a new one first. Then, select the group, building system, distribution system, or zone to add to or remove from the subgroup.

7.69.2 Options within the command

List groups
Lists all existing groups in the Command line.

Create a group
Creates a new group to which nested subgroups can be added later.

Group
Adds the new group as a general BIM Group.

Building system
Adds the new group as a BIM Building System.

Distribution system
Adds the new group as a BIM Distribution System.

Zone
Adds the new group as a BIM Zone.

liSt sub groups
Lists all existing subgroups in the Command line.

Remove subgroup
Removes the subgroup.

7.70 BIMZONE command

Groups Spaces into a zone.

Shape Lite Pro Mechanical BIM

7.70.1 Method

Enter an existing zone name or a name for a new zone, then select the entities (Spaces) to add to or remove from the zone.

Note: Zones created with BIMZONE command are exported with IFCEXPORT command as IfcZone.

7.70.2 Options within the command

List zones
Lists all existing zones in the Command line.

Show properties
Allows you to edit the properties. The available properties are displayed in the Command line.

Ifc properties
Allows you to edit the IFC properties. The available IFC properties are displayed in the Command line.

Delete zone
Deletes the selected zone.

Create a zone
Creates a new zone.

7.71 BIMSETREFERENCEFACE command

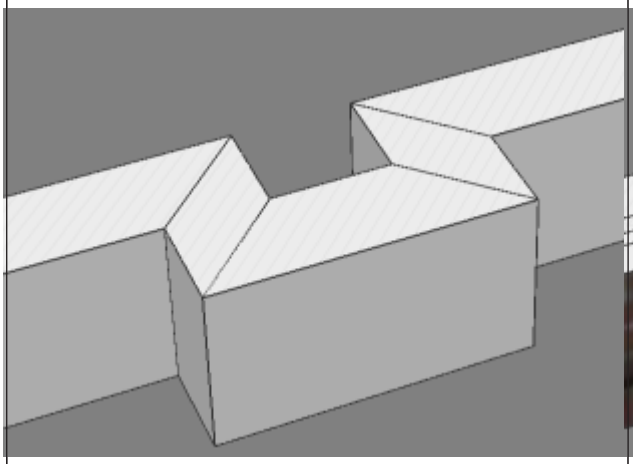
Defines a reference and an opposing face to control the ply layout of a composition.

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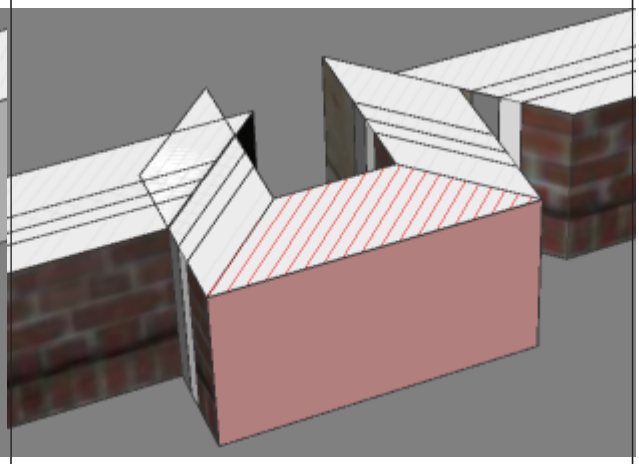
7.71.1 Methods

The reference face can be set manually by selecting a face. If the opposing face is parallel to the reference face, a suggestion for the opposing face is given and can either be accepted or be changed.

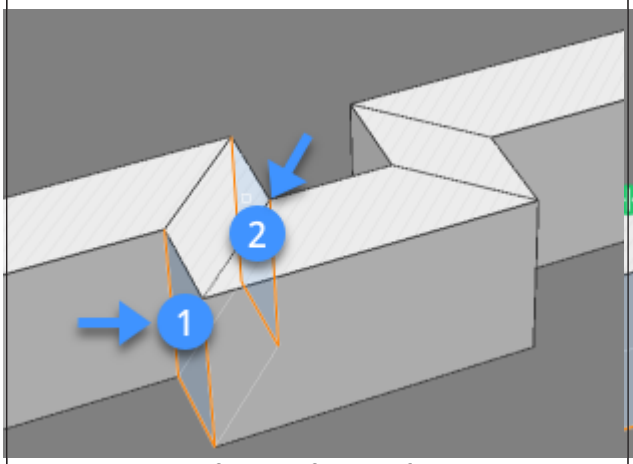
Note: If one of the plies of the composition has a variable thickness, the reference and opposing face can be non-parallel. In this case the fixed thickness plies are set out starting from the reference face, and the remainder of the solid is filled by the variable ply.



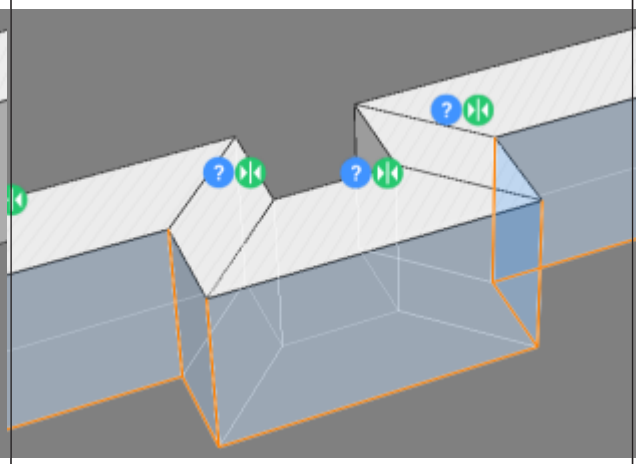
Solids without composition



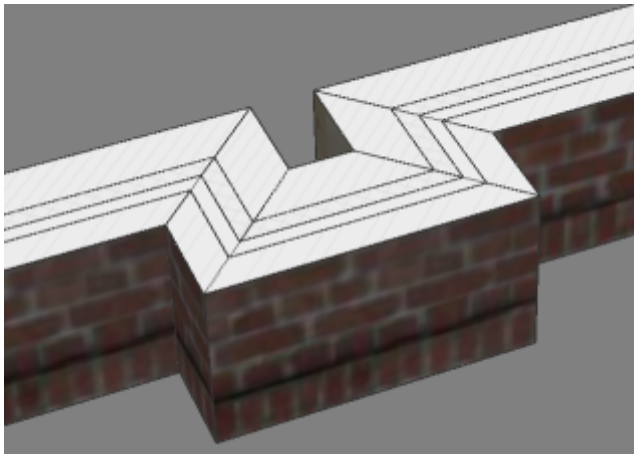
Multiply composition assigned
Reference faces are either wrong or not found



Defining reference face
(1) and opposing face (2)



Attaching a composition
Reference faces highlight



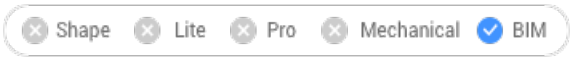
Result

7.71.2 Options within the command

Select manually
Selects the opposite face manually.

7.72 BIMSLAB command

Creates solids classified as **Slab**.



Icon:

7.72.1 Description

The command creates slabs based on a closed 2D curve or boundary. You can define the options through the command context panel, as well as through the Command line.

7.72.2 Method

Launch the command to open the **Slab** command context panel.

There are three methods to define the out limits of the slab:

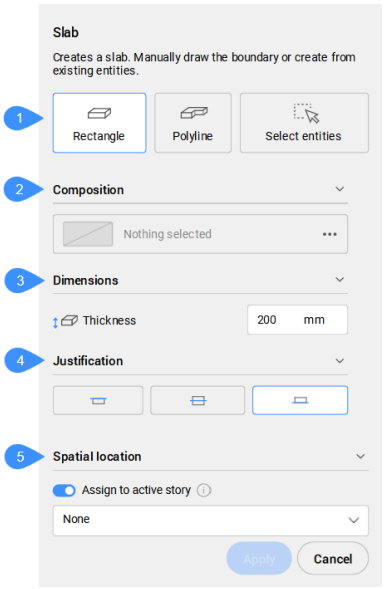
- Draw a rectangle.
- Draw a polyline.
- Select entities in the drawing.

Use the **Hotkey Assistant** widget to change the justification of the current slab. Press the **Ctrl** key to switch between the displayed options.



Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMSLAB options** check box is ticked in the **Hotkey Assistant Configuration** dialog box (see **Hotkey Assistant widget** article).

7.72.3 Options within the command context panel



1 Creation mode

2 Composition

3 Dimensions

4 Justification

5 Spatial location

Creation mode
Allows you to choose a method for creating the slab.

Rectangular
Defines the new slab boundary by manually drawing a rectangle.

Polyline
Defines the new slab boundary by manually drawing a polyline.

Select entities
Allows you to select an entity from the drawing to define the new slab boundary. You can select either a single 2D curve/region/boundary, or multiple walls. When these entities are preselected before launching the BIMSLAB command, the panel automatically selects this mode.

Composition
Opens the **Compositions** dialog box that lets you define the current slab composition. By default the compositions of type **Slab** are shown. You can change this filter in the **Compositions** dialog box.

Dimensions

Thickness
Sets the slab thickness.

Note: When a composition with a fixed thickness is selected, this parameter is greyed out.

Justification
There are three justification options: top, center and bottom. By default, the justification is set to bottom. To change it, click another justification option.

Spatial location
Allows you to select a spatial location from the drop-down menu to assign to the slab.

Assign to active story
Allows you to copy the spatial location of the active story which is currently active in **TVM** or of the closest story level if not in **TVM**.

7.72.4 Options within the command line

Offset
Specifies the offset distance of the slab's out limits through point selection or by typing a number in the dynamic dimension field.

7.73 BIMSLICEWALL command

Slices a selected wall vertically.

Shape Lite Pro Mechanical BIM

Icon:



7.73.1 Method

Select a wall, a wall-like solid or a linear solid. The BIMSLICEWALL command automatically defines the plane as a vertical plane, perpendicular to the length direction of the solid.

A blue line displays at the cursor, from the cursor to both ends of the solid, which allows to precisely indicate the slicing position dynamically.

7.74 BIMSPACE command

Creates BIM Space entities from enclosed boundaries.

Shape Lite Pro Mechanical BIM

Icon: 

7.74.1 Description

Creates BIM Space entities by picking a point inside an enclosed area defined by space bounding entities.

Note: Space bounding entities are 3D solids or linear 2D entities, which are classified as BIM entities and have the **Space bounding** property set to **On**. If the **Centerline** property of a space bounding solid is set to **On**, the center of the solid is used as the space boundary.

A label is created at the geometric center of the space.

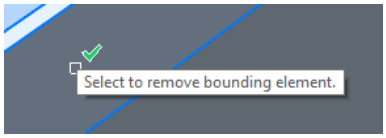
Note: The **Space** option of the BIMCLASSIFY command converts any 2D or 3D entity into a BIM Space entity.

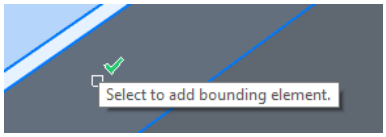
The height of the space is defined by:

- The top face of the highest bounding entity.
- The **Default Space Height** user preference, when all bounding entities are 2D linear entities.

7.74.2 Options within the command

Edit
Launches the **Edit** mode, which allows you to add or remove space bounding entities in a Space by clicking them.





Note: After editing, launch the BIMUPDATESPACE command to update the space.

Note: If the **Update method** property of the space is set to **Manually**, the space cannot be updated by the BIMUPDATESPACE command. This can be solved by setting this property to **Automatic**.

Note: The BIM Spaces properties can be edited in the **Properties** panel.

BIM Space Properties

Representation
Sets the visual representation of the BIM Space:

- **Footprint**: Space stamp only.
- **Solid**: Transparent 3D solid. The height of the solid is copied from the surrounding wall solids.

Status
Specifies the status of the Space.

Note: If the space is out-of-date, an exclamation mark icon is displayed next to the space stamp. This can be solved with the BIMUPDATESPACE command.

Name
Sets the name of the space, which is also visible on the space stamp.

Description
Describes the space.

Building
Specifies the building to which the space is assigned to.

Story
Specifies the story to which the space is assigned to.

Number
Assigns a new number to the space. By default the space is automatically numbered.

Interior Elevations
Controls the display of **Interior Elevations**. If interior elevations are not created yet, the **Interior Elevations** property is disabled.

Entity property sets
Click the **Browse** button to open the **Enable per-instance property set** dialog box, then check the **User** namespace.

Note: Additional properties can be created using the BIMPROPERTIES command.

Note: Assigning a **Space** property to a building element overrides the element's **Story** and **Building** properties with those of the space.

7.75 BIMSPATIALLOCATIONS command

Creates and edits site, buildings and stories.

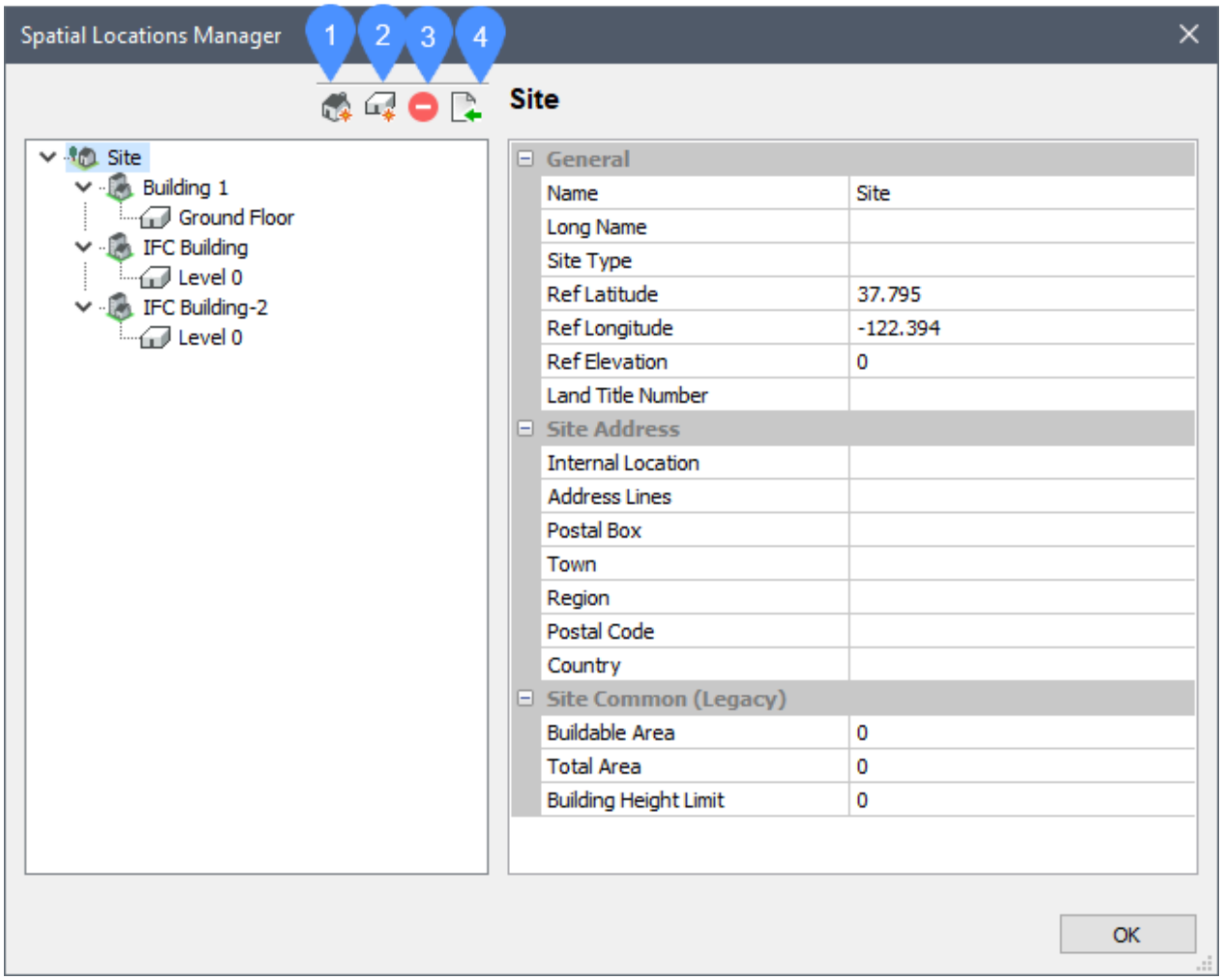
Shape Lite Pro Mechanical BIM

Icon: 

7.75.1 Description

Displays the **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

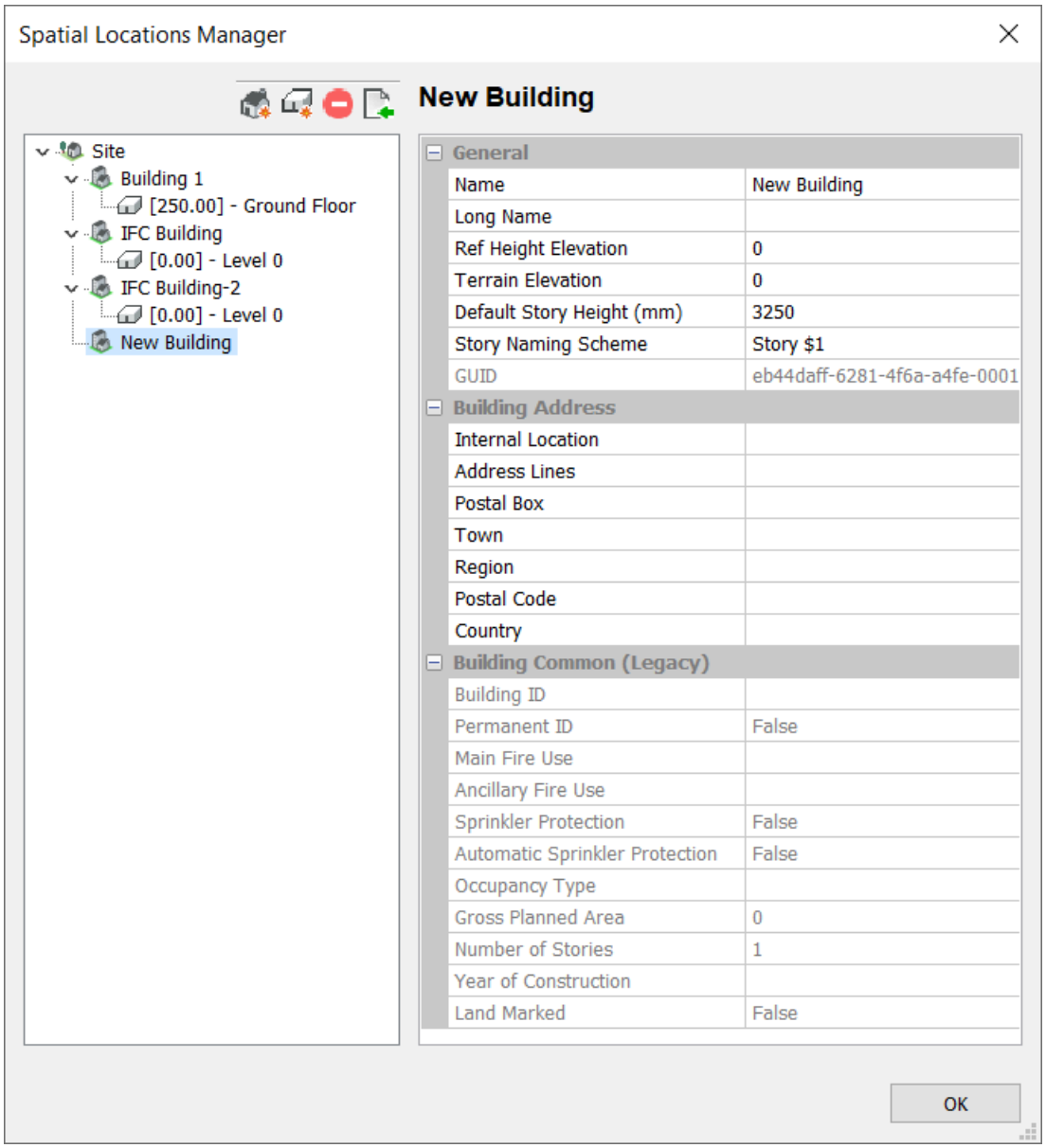
3 Delete selected Building or Story

4 Import spatial locations

7.75.2 New Building

Adds a new building to the model.

Fill out the properties grid:



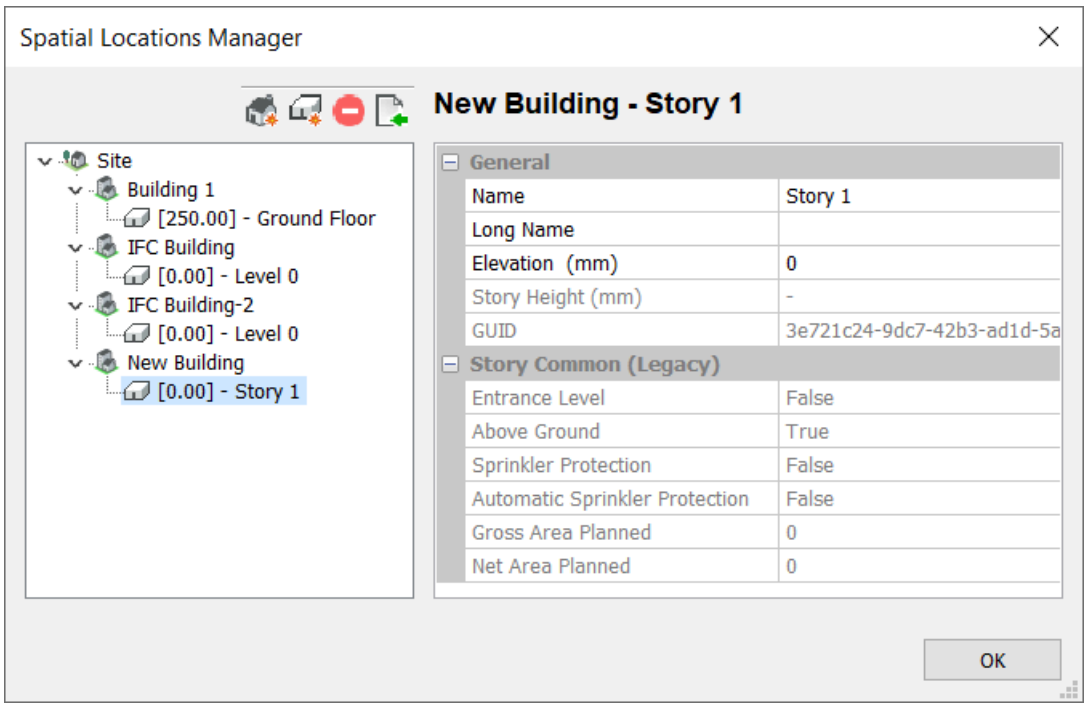
Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.
- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

7.75.3 New Story

Adds a new story in the selected building.

The properties grid is completed.



7.75.4 Delete selected Building or Story

Deletes the selected building and story.

7.75.5 Import spatial location

Imports spatial locations from a text file (.txt).

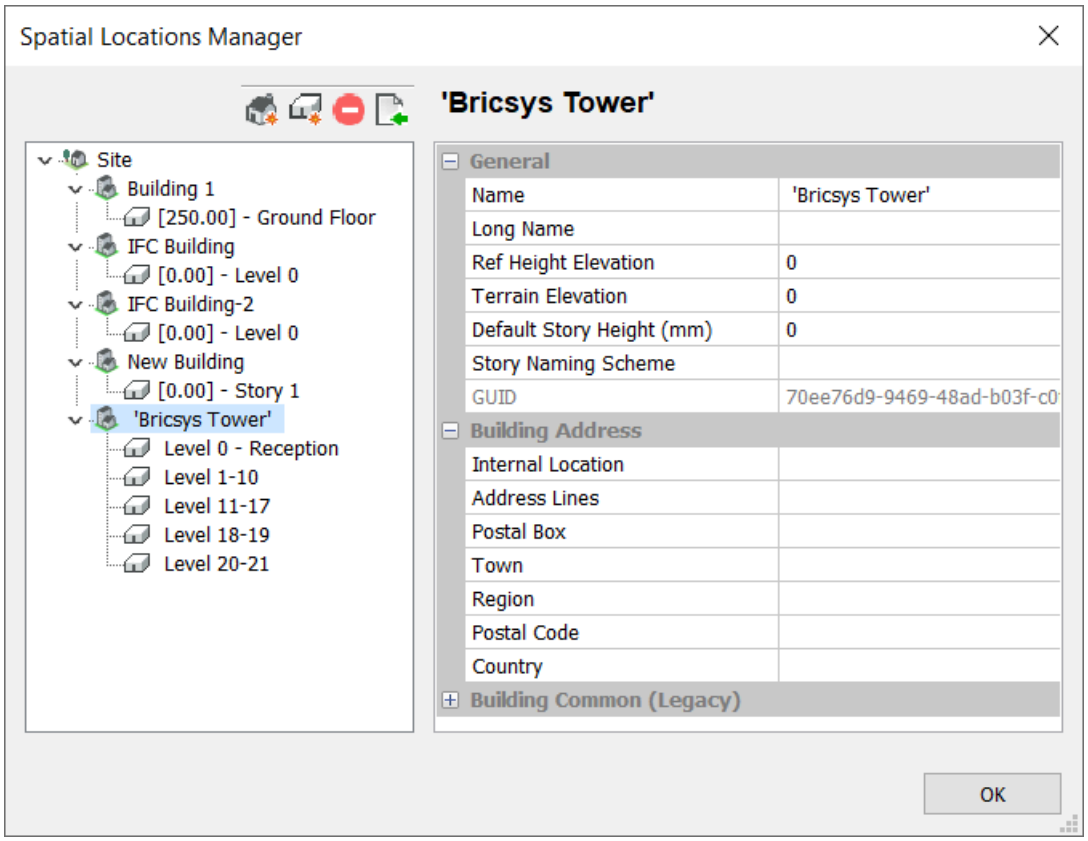
Displays a **Standard Open Files** dialog box from which will open the *.txt spatial location file.

Example of a *.txt spatial location file:

```
Space:Name= 'Bricsys Tower' ,Area=15000 Space:Name= Level 0 - Reception ,Area=6%,
 ' Floor-to-Floor Height '=5000, Count=1 Space:Name= Level 1-10 ,Area=4%, ' Floor-
to-Floor Height '=4000, Count=10 Space:Name= Level 11-17 ,Area=7%, ' Floor-to-
Floor Height '=3000, Count=7 Space:Name= Level 18-19 ,Area=5%, ' 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.

The spatial location file will be loaded as below:



7.75.6 Options within the command

Dialog
Opens the **Spatial Locations Manager** dialog box.

add Building
Adds a new building to the model.

Rename Building
Renames a building.

add Story
Adds a new story in the selected building.

Rename Story
Renames a story.

sEt story elevation
Sets an elevation for an existing story by entering the elevation value or select a point.

Point
Sets the point for story elevation.

7.76 BIMSPLIT command

Splits segmented solids into separated solids.

Shape Lite Pro Mechanical BIM

Icon:



7.76.1 Description

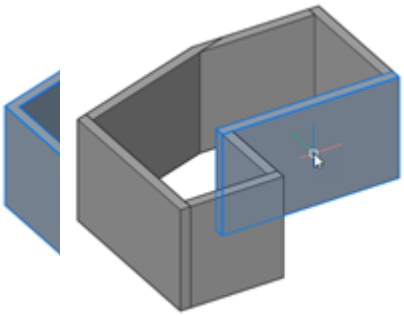
Automatically splits segmented solids into separate solids and allows to split a solid using a selection of cutting faces. Specifically designed to recognize planar & linear shapes such as walls, slabs & columns it can help you split your mass model into separate entities, which is necessary for a typical BIM workflow.

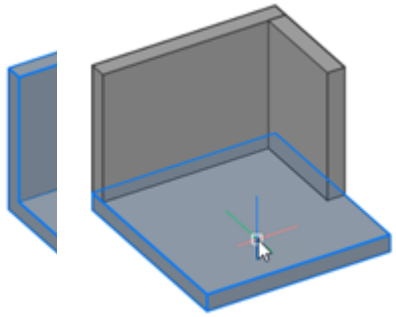
7.76.2 Method

Select a solid for auto splitting

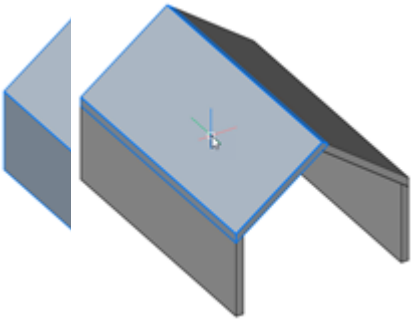
One method is to select an entire solid and let BIMSPLIT decide automatically where the solid should be cut. This will be done along logical cutting faces, yielding results as shown in the table below.

Note: For complex geometry BIMSPLIT may not give the expected result. In that case, try simplifying by first splitting the geometry into smaller parts manually.

Solid selected…	**…split into 7 separate solids**
	
Solid selected…	**…split into 3 separate solids**



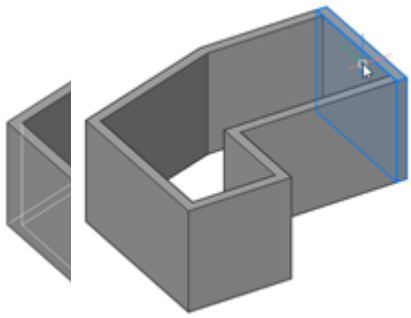
Solid selected ... **...split into 4 separate solids**



Select cutting faces for manual splitting

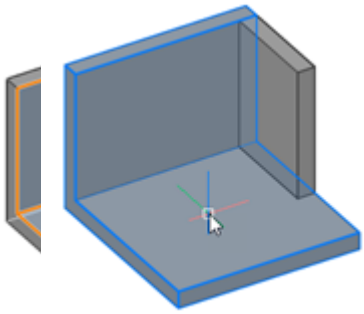
Another method is to select cutting faces manually. This behavior is similar to the SLICE command, the main difference being that it is possible to select multiple cutting faces. Adjacent cutting faces will keep their respective parts of the solids intact.

One cutting face selected ... **... split into 2 separate solids**



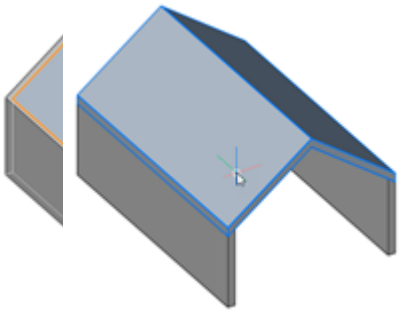
Two cutting faces selected...

...split into 2 separate solids



Two cutting faces selected...

...split into 3 separate solids



Note: Inserts, such as windows and doors, which existed in the split solid are removed.

7.77 BIMSTAIR command

Creates stairs.

Shape | Lite | Pro | Mechanical | BIM

Icon:

7.77.1 Description

Creates stairs between two floor slabs or any empty spaces. This command is using the distance between the Z-values of the selected start plane and end plane as height.

The stairs display automatically when the cursor is moved. The type of stair being created depends on the position of the cursor with respect to the position of the selected start point.

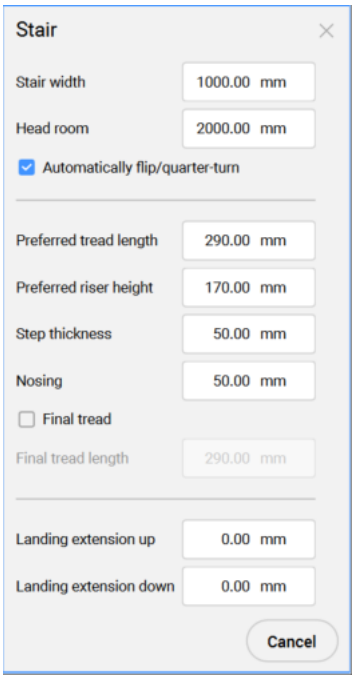
Note: The staircase is created as a parametric block. The Stair Type is assigned automatically and can be edited in the **Properties** panel. Editing this property does not affect the geometry of the staircase.

7.77.2 Method

There are two methods to create stairs using the BIMSTAIR command:

- Select a horizontal face of a solid.
- Select a void area.

The **Stair** command panel opens, which allows you to easily change the settings while placing the stair.



Stair width
Sets the stair width of the stairs.
The default value is 40 for MEASUREMENT=0 (inches), and 1000 for MEASUREMENT=1 (millimeters).

Head room
Sets the minimal head clearance of the stairs.
The default value is 80 for MEASUREMENT=0 (inches), and 2000 for MEASUREMENT=1 (millimeters).

Automatically flip/quarter-turn
Toggles the stairs type.

Preferred tread length
Sets the preferred tread length of the stairs.
The default value is 11.5 for MEASUREMENT=0 (inches), and 290 for MEASUREMENT=1 (millimeters).

Preferred riser height
Sets the preferred riser height of the stairs.
The default value is 7 for MEASUREMENT=0 (inches), and 170 for MEASUREMENT=1 (millimeters).

Step thickness
Sets the thickness of the individual steps.
The default value is 2 for MEASUREMENT=0 (inches), and 50 for MEASUREMENT=1 (millimeters).

Nosing
Sets the horizontal distance of the overlap between subsequent treads.
The default value is 2 for MEASUREMENT=0 (inches), and 50 for MEASUREMENT=1 (millimeters).

Final tread
Controls the creation of a tread at the elevation of the top bounding slab.

Final tread length
Sets the length of the final tread.

The default value is 11.5 for MEASUREMENT=0 (inches), and 290 for MEASUREMENT=1 (millimeters).

Landing extension up
Sets the extension of the landings created with the BIMSTAIR command in the **up** direction. Default value is 0.

Landing extension down
Sets the extension of the landings created with the BIMSTAIR command in the **down** direction. The default value is 0.

Note: The last value for the above settings are remembered.

Note: For all properties, parameters will be created when creating the stair, driving the constraints of the parametric stair block. These parameters can be changed individually afterward in the **Parameters** section of the **Properties** panel.

7.77.3 Options within the command

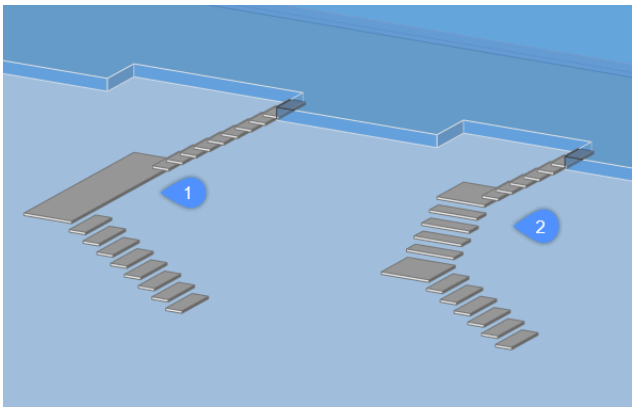
Quarter-turn
Turns the direction of the stair cursor by 90 degrees.

Note: Alternatively, you can press the **Ctrl** key once to turn the stair cursor by 90 degrees.

Flip
Turns the direction of the stair cursor by 180 degrees.

Note: Alternatively, you can press the **Ctrl** key twice to turn the stair cursor by 180 degrees.

U-shape type
Switches between two different U-shape type stairs:

- Double-flight U-shaped stairs (1): with a single rectangular landing solid bridging the perpendicular distance between both opposing stair flights.
- Triple-flight U-shaped stairs (2): with two square landing solids between each pair of subsequent stair flights.
- 

Note: Press the **Ctrl** key to switch between the possible stair types with respect to the current position of the stair cursor.

Note: To use the **Ctrl** key functions, the HOTKEYASSISTENT must be set to **On**.

7.78 BIMSTRETCH command

Stretches the ends of linear solids.

Shape Lite Pro Mechanical BIM

7.78.1 Description

Stretches the ends of linear solids while maintaining previously made connections between other linear solids in the overall structure.

7.78.2 Method

There are two methods to use BIMSTRETCH command:

- Stretch structural connections.

Note: Stretching elements that are classified as Beam, Column or Member.

- Stretch MEP connections.

Note: Stretching elements that are classified as Flow segments.

Before using the BIMSTRETCH command make sure that:

- 'Display sides and ends' is enabled.
- 'Display Axes' is enabled.
- Structural profiles are connected with StructuralConnect.
- MEP connections are connected with FlowConnect.

Note: Before running the command, select the ends of the entities that will be modified.

7.78.3 Options within the command

Base Point
Specifies new basepoint to start the displacement from.

Copy
Makes a copy of the connected entities and places them in the new position.

Undo
Undoes the last action.

Exit
Exits the command.

7.79 BIMSTRUCTURALCONNECT command

Connects structural profiles.

Shape Lite Pro Mechanical BIM

Icon: 

7.79.1 Description

Connects structural profiles such as beams, columns and members.

7.79.2 Method

There are two methods to use BIMSTRUCTURALCONNECT command:

- Select two solids.
- Select multiple solids.

Note: The axes of the solids must be coplanar.

7.79.3 Options within the command

Select base solid

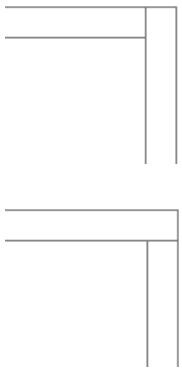
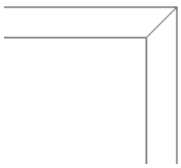
Specifies the base solid.

Note: This option is only available for L-connection.

Switch

Allows to choose between different types of L-connections:

- Butted connection
- Mitered L- connection
- Touching/disconnected connection

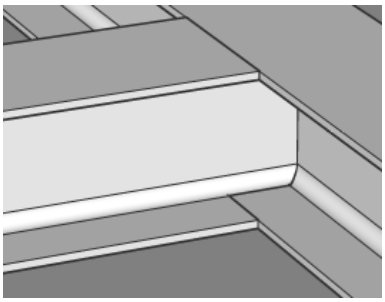
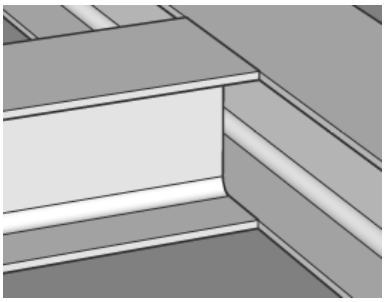
Butted Connection	Mitered L-connection	Touching connection
		

Note: Press the Ctrl key to cycle between the connection types. Hot Key Assistant must be set ON.

Note: If a T-connection is created, there are no options available.

Connection cut type

Depending on the DMCONNECTIONCUTTYPE system variable value, the connection type can be smooth or planar.

Smooth connection	Planar connection
	

7.80 BIMTAG command

Creates associative tags.

Shape Lite Pro Mechanical BIM

Icon:

7.80.1 Description

Creates associative tags in generated BIM section drawings for the corresponding building elements in a 3D BIM model. The tags will have an **Associativity** property and an **Associative coloring** property. The **Associativity** property indicates whether a tag is associated to a BIM element. The **Associative coloring** property visualizes an unassociated tag in red, if the property value is **On** (this doesn't affect the entity's **Color** property).

Note:

- This command can only be used on section viewports.
- The GENERATEASSOCVIEWS system variable must be **ON** when generating or updating tags.

7.80.2 Method

The tags are created automatically for the selected building elements.

Note:

- Tags are created as multileader entities using multileader styles in _SectionTag.dwg. The content of such a multileader style can be a field refering to a property of BIM entity type or a block with attributes. Each attribute refers to a property of a BIM entity type. BIM entity types are linked to a multileader style in _TagTypeToStyle.xml. All source files can be found in the Support folder, for example: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections*.
- Detail tag is added when a detail reference in the model is sectioned.
- Section indicators, story indicators and grid call-outs are generated automatically within section update.

- Grid and Space call-outs are stored in the _SectionSettings.dwg file.

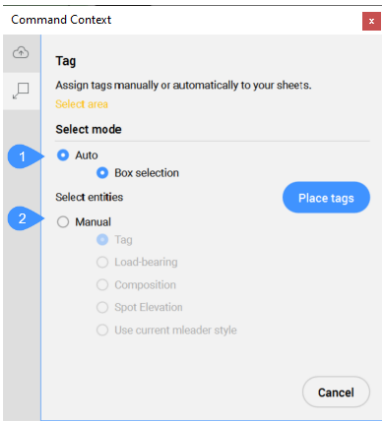
The selection modes are:

- **Automatic**: selects a viewport.
- **Box selection**: selects an area of a viewport.
- **Manual**: individually selects entities in the viewport by picking a point on their surface.

  Note: If the selected point is shared between multiple BIM entities, use the **Ctrl** key to cycle through the different possible tags.

Note: Both **Manual** and **Box selection** modes allow you to tag background entities.

The command opens the **Tag** command context panel to let you access the options and see useful messages and instructions.



1 Auto

2 Manual

Auto
Creates tags automatically for all sectioned building entities in the selected viewport, including section indicators, story indicators, and grid callouts.

Box selection
Creates tags for all entities in a selected area of viewport. After selecting the area, click the **Place tags** button to have the tags created in the viewport.

Manual
Creates a tag with the selected tag type for the selected element in a viewport.

Tag
Allows you to pick a point on the element, then drag the tag to adjust its position and left-click to place it.

Load-bearing
Inserts a load-bearing direction symbol when a generated line of a slab solid is selected. Make sure the load-bearing direction is set by the BIMSETLOADBEARINGDIRECTION command.

Composition
Tag with all materials of composition of the sectioned BIM element.

Spot Elevation
Inserts a tag that indicates the elevation of the identified point in elevations and vertical sections.

Note: The following options are available only if tags have already been assigned to the section.

Use current mleader style
Allows to use the current multileader style instead of the default style for selected sectioned BIM elements.

Note:

- Both **Manual** and **Box Selection** modes allow you to tag background entities.
- If the selected point is shared between multiple BIM entities, use the **Ctrl** key to cycle through the different possible tags.
- When using projects created with BricsCAD V23 in newer BricsCAD versions and adding new sheets alongside sheets from V23 that already contain the IFC2x3 tag definitions inside the DWG file sheets, some default BIM tags (_DoorTypeStyleMax, _WindowTypeStyleMax, and _StairTypeStyleMax) will not work correctly.

  To be able to use BIMTAG command in this situation, you should:

  - **in Manual mode**: use the CTRL to loop through the different tags until the IFC2x3 compliant tag is met.
  - **in Auto mode**: in case you want to use the IFC2x3 tags, update the _TagTypeToStyle.xml (located in *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections*) by setting the **autoPlacement** value to **true** for the IFC2x3 tags, and **false** for the others.



The options within the command context panel can also be accessed from the Command line.

7.80.3 Options within the command line

change Tag type
Changes the type of the tags to be created.

Note: The following options are available only if tags have already been assigned to the section.

Update existing
Updates the existing tags in the selected viewport. The position of relocated tag blocks is preserved.

update existing and add New
Updates the existing tags in the selected viewport and adds new tags.

Regenerate all
Updates all tags in the selected viewport. Repositioned tag blocks are moved to their default location.

7.81 BIMTRIM command

Trims a wall.

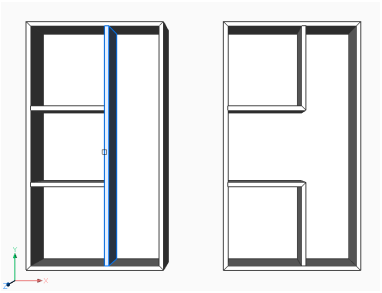
Shape Lite Pro Mechanical BIM

Icon:

7.81.1 Method

The command trims a part of a wall by automatically detecting other walls and using them as trim objects.

Select wall
Select the part of the wall that should be trimmed.



7.82 BIMUPDATESPACE command

Recalculates a BIM space which is out-of-date.

Shape Lite Pro Mechanical BIM

Icon:

7.82.1 Description

Recalculates a BIM space which is out-of-date, for example, after adding or removing bounding entities. After the command is run, select all spaces to update.

Note: Only spaces that have the **Update method** property set to **Automatic** can be processed.

7.83 BIMUPDATETHICKNESS command

Re-applies the overall thickness of a composition to the solid.

Shape Lite Pro Mechanical BIM

Icon:

7.83.1 Method

Select the solids to re-apply the overall thickness of composition to.

The command finds the broken connections (mitered/butted) between the selected solids and adjacent solids and automatically restores them.

Note: Make sure the KEEPCONNECTIONS system variable is turned On.

Accept/reject all restored connections or check connections individually.

Note: Only solids that need to be updated are processed.

7.83.2 Options within the command

Symmetrical Update
Updates the thickness symmetrically or with respect to the reference face. See also the **BIMATTACH-COMPOSITION command** article.

Note: If the thickness is updated symmetrically, the position of the axis of the solid is preserved.

Accept
Accepts all automatically restored wall-wall connections.

Reject
Rejects all automatically restored wall-wall connections.

Check individually
Zooms in the view to each restored connection and lets you accept/reject it.

Yes
Accepts the restored connection. Select **Confirm**.

No
Rejects the restored connection. Select **Confirm**.

Confirm
Confirms the previous option selection.

7.84 BIMUPDATESTORYHEIGHT command

Changes the height of a story in a BIM model.

Shape Lite Pro Mechanical BIM

Icon: 

7.84.1 Method

Launch the command to open the **Update Story Height** command context panel.

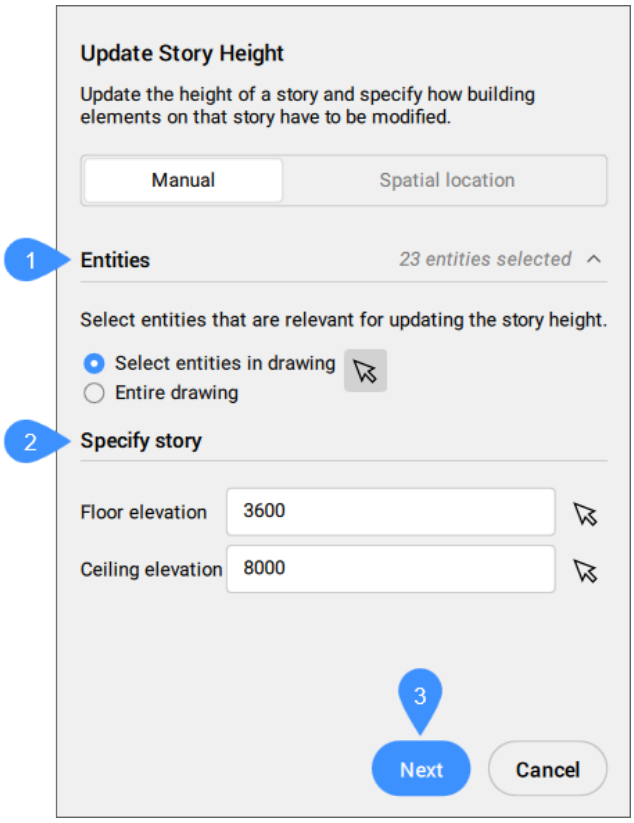
Define where in the model the change in height needs to be applied. This can be done via the **Spatial Locations** tab by selecting a story from the list. In this case, the current elevation levels will be determined based on the selected Story, and all entities that are not contained in the Building of the selected Story, will be ignored when updating the story height.

Use the **Manual** tab for manually selecting two elevations in your model and/or creating a custom set of building elements that are relevant for updating the story height.

Define a new story height and select operations for each entity or entity type. Choose whether the entities should change height, move with the story height change, or stay intact.

Preview the changes. You can further make changes or accept the results.

7.84.2 Manual tab



1 Entities

2 Specify story

3 Next

Entities
Selects the building elements that are relevant for updating the story height.

Select entities in the drawing

Allows you to manually select entities to be modified by the story height update. Click the **Select** arrow ( ) to start creating the selection set.

Note: Make sure to not only select entities on the story for which the height has to change, but also the entities above, because they will need to be moved.

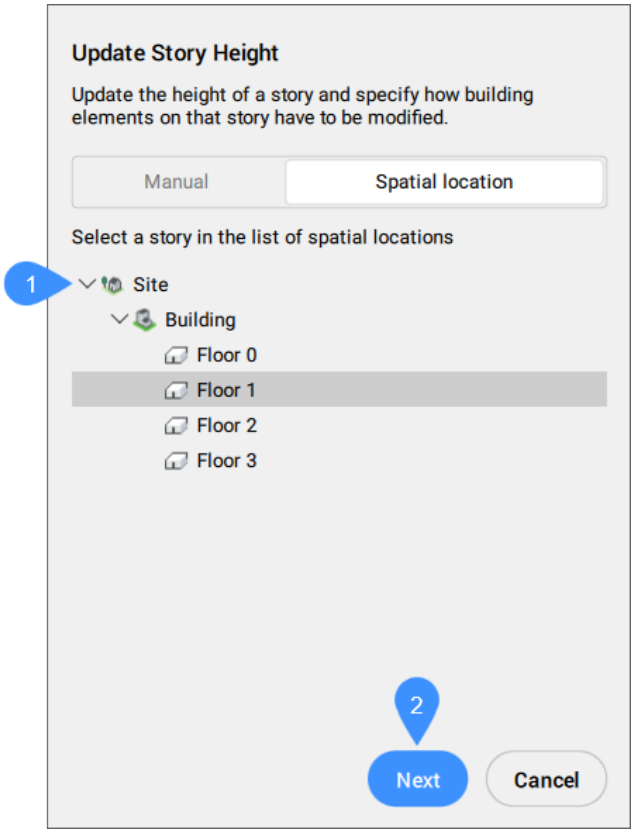
Entire drawing
Analyzes the entire drawing and automatically detects the entities that are relevant for the selected story.

Specify story
Allows you to define the **Floor elevation** and **Ceiling elevation** by typing the elevations values in the input fields or by using the **Select** arrow ( ) to pick points in the drawing. The floor elevation and ceiling elevation of the selected story are highlighted in the model by two green planes.

Note: A warning message is displayed if the defined values are incorrect.

Next
Displays the options related to the selected level.

7.84.3 Spatial location tab



1 Spatial locations

2 Next

Spatial locations

Lists all the BIM model's buildings and their stories.

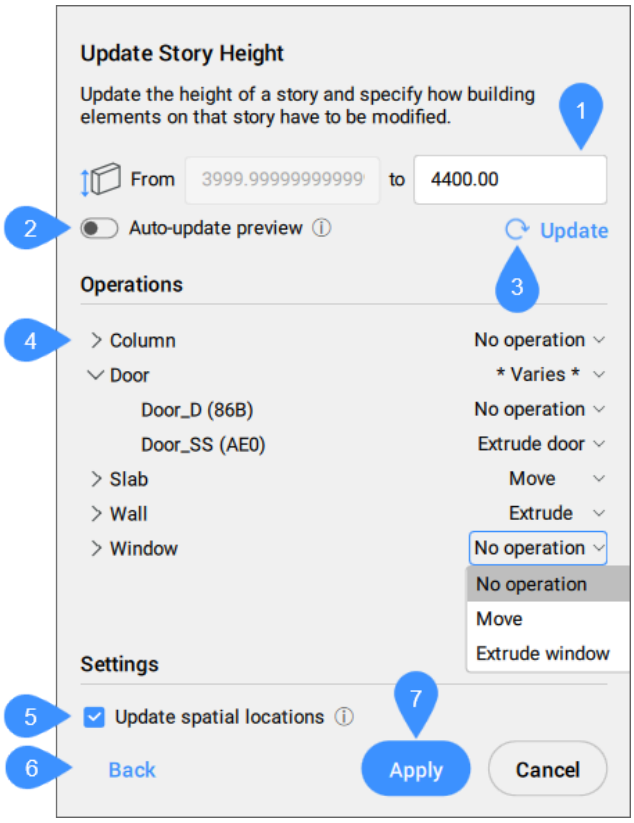
Select the story for which you want to update the height. The floor elevation and ceiling elevation of the selected story are highlighted in the model by two green planes.

Note: In the Command line, the **Spatial locations** option allows you to enter the building and story numbers.

Next

Displays the options related to the selected level.

7.84.4 Preview operations



1 New story height

2 Auto-update preview

3 Update

4 Operations

5 Update spatial locations

6 Back

7 Apply

New story height

Allows you to type in a new value for the story height.

Note: In the Command line, the option is **set new story Height**.

Auto-update preview

Turns on/off the preview automatic update after each modification. Enabling the **Auto-update preview** option disables the **Update** option.

Note: Preview updates may be slow if there are many entities or if there is complex geometry involved.

Update

Allows you to manually update the preview after several modifications.

Operations

Lists the building elements that can be affected by an operation, grouped by their BIM type. Selecting entities in the list highlights them in the drawing (and vice-verse).

A menu with operations is displayed at the right hand-side of each entity type/entity: **No operation**, **Move**, **Extrude**, **Extrude door**, **Extrude window**, **Scale block**.

Note: Only blocks with an axis parallel with the model's Z-axis are scaled (the geometry inside the block definition is not changed).

Select an operation to be applied to all the entities in a group, expand the group and select operations for each entity, or keep the suggested operations.

Update spatial locations
Toggle the update of spatial locations when the story height update is applied.

Note: This option is available only for the **Spatial location** mode. In the Command line, you can find it under **Settings** > **Spatial locations**.

Back
Returns to the selection tab.

Apply
Applies the changes to the model.

Note: The options within the Command line reflect the options within the command context panel.

7.85 BIMWALL command

Creates solids classified as **Wall**.

Shape Lite Pro Mechanical BIM

Icon: 

7.85.1 Description

The command allows you to easily create walls from scratch or detect walls from point clouds, aligned with UCS X- or Y-axis or unconstrained by these. You can define the options through the command context panel, as well as through the Command line.

7.85.2 Method

Launch the command to open the **Wall** command context panel.

Note: To place walls more easily, enable **Top View Mode** (**TVM**) before launching the command by clicking a story disk () in the **Story Bar** (see **The Story Bar** article).

There are three methods to create walls:

- Place single walls, constrained by the X-/Y- axis, or by a fixed angle.
- Draw poly walls, unconstrained by the X-/Y- axis.
- Select entities in the drawing to extrude as walls.

In **Single wall** mode, the current wall automatically extends to any wall that is visible in the current view. The result can thus be a wall with:

- both ends attached to other walls.

- only one end attached to another wall, while you can dynamically specify the length of the current wall.
- none of the ends attached to other walls, while you can set a value in the **Free wall length** field.

Use the **Hotkey Assistant** widget to change the direction of the current single wall. Press the **Ctrl** key to switch between the displayed options.



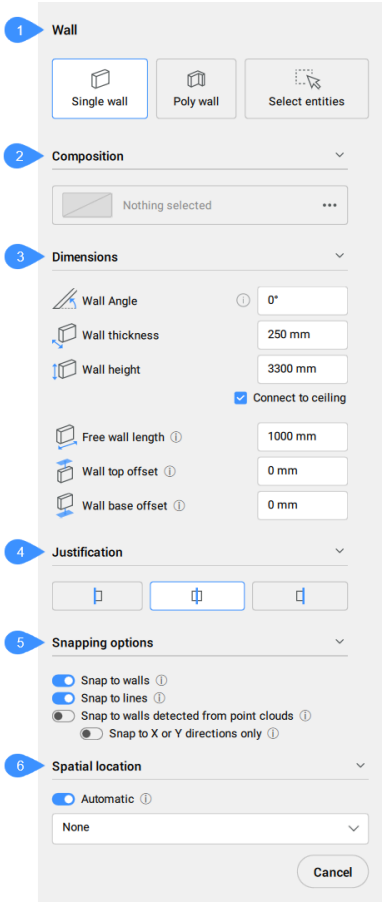
Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMWALL options** check box is ticked in the **Hotkey Assistant Configuration** dialog box (see **Hotkey Assistant widget** article).

Use the dynamic dimensions to define the insertion point more accurately. These dimensions show the distances from the current single wall to other, parallel, walls. Tap the **Tab** key to switch between dimensions and set them manually.

Note: Dynamic dimensions are displayed if **Dynamic Input** (**DYN**) is set to **On** (see **Dynamic dimensions** article).

Use the selection dimensions to reposition the selected wall relative to the nearest detected walls and/or columns or other convenient reference points (see **Selection dimensions** article).

7.85.3 Options within the command context panel



1 Creation mode

2 Composition

3 Dimensions

4 Justification

5 Snapping options

6 Spatial location

Creation mode

Allows you to choose a method for creating the current wall/walls.

Single wall

Creates X-aligned or Y-aligned single walls.

Poly wall

Creates a polyline shaped wall, unconstrained by the X- or Y-axis. You can define the new wall by manually drawing a polyline.

Select entities

Allows you to select polylines in the drawing from which to create the new walls.

Composition

Click the browse button (...) to open the **Compositions** dialog box that lets you define the current wall composition. By default, the compositions of type **Wall** are shown. You can change this filter in the **Compositions** dialog box.

Dimensions

Allows you to define the dimensions of the current wall.

Wall angle

Sets the direction of the current wall. Only available for **Single wall** mode.

Wall thickness

Sets the thickness of the current wall.

Note: When a composition with a fixed or minimal thickness is selected, the thickness input field is constrained accordingly.

Wall height

Sets the height of the current wall.

Connect to ceiling

Automatically fills in the **Wall height** field by searching for solids above the wall to connect to. When no solids are above the created wall, the default **Wall height** will be used.

Free wall length

Sets the length of the current free wall. Only available for **Single wall** mode.

Wall top offset

Sets the top offset from the slab detected above the current wall.

Note: This option becomes unavailable when the **Connect to ceiling** option is set to **Off**.

Wall base offset

Sets the base offset from the slab detected below the current wall.

Justification

There are three justification options: left, center and right. By default, the justification is set to center. To change it, click another justification option.

Snapping options

Toggles **On/Off** multiple wall snapping options. Only available for **Single wall** mode.

Snap to walls

Snaps the wall cursor to existing walls and wall extensions.

Snap to lines

Snaps the wall cursor to existing lines, both single and double. For double lines, their distance will be adopted as the wall thickness.

Snap to walls detected from point clouds

Snaps the wall cursor to nearby walls detected from an underlying point cloud and adopts their thickness and direction.

Note: When snapping to external walls, the thickness of the current wall is defined by the value specified in the **Wall thickness** field.

Snap to X or Y directions only
Constrains the snapping directions of the current wall to X- or Y- axis.

Note: The order in which snapping options are present in the command context panel represents their application hierarchy.

Connections

Connect to snapped walls
In **Poly wall** mode, controls whether the Poly wall's start and end will be connected to snapped walls (mitered connection).

Spatial Location
Accept the automatically assigned spatial location or define a new one by selecting an option from the drop-down menu.

Automatic
Copies the spatial location of the closest underlying slab.

Note: Selecting a different spatial location than the one automatically assigned toggles the **Automatic** option to **Off**.

Note: The options within the command context panel and those within the **Hotkey Assistant** widget reflect the options within the Command line.

7.86 BIMWINDOWCREATE command

Creates a fully parametric window or opening based on a closed contour.

Shape Lite Pro Mechanical BIM

Icon: 

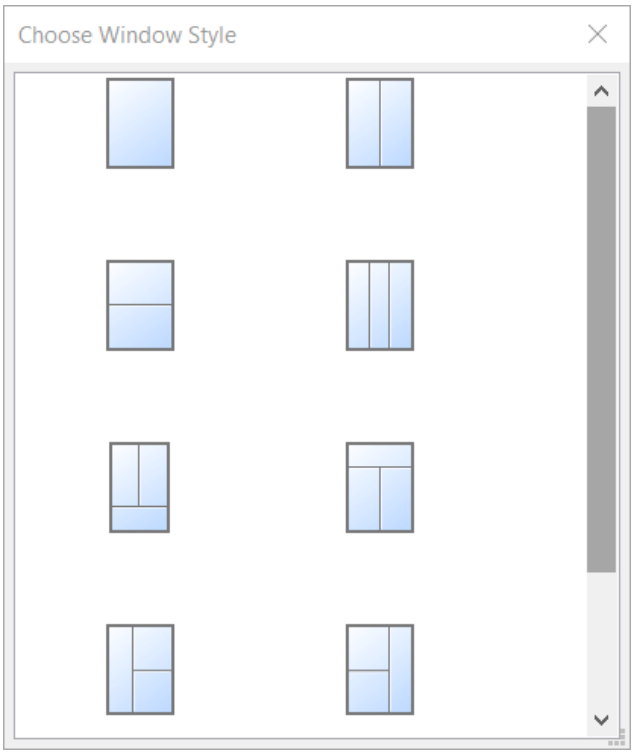
7.86.1 Description

Creates parametric windows using as profile any entity such as a boundary, a closed polyline or grid.

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

7.86.2 Options within the command

Window
Creates a BIM Window entity. An opening in the solid on which the profile entity lies is created automatically. The **Choose Window Style** dialog box opens and the windows style can be chosen.
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.



Opening

Creates an opening in the solid on which the profile entity lies. The opening is created by a solid on the **BC_SUBTRACT** layer. This layer is frozen by default. Thaw the layer to select the opening.

7.87 -BIMWINDOWCREATE command

Creates a fully parametric window or opening based on a closed contour.

Shape Lite Pro Mechanical BIM

7.87.1 Description

Creates parametric windows using as profile any entity such as a boundary, a closed polyline or grid.

Note: This command operates at the Command line.

7.87.2 Options within the command

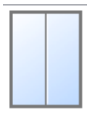
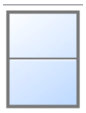
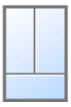
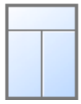
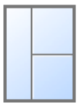
wIndow

Creates a BIM Window entity. An opening in the solid on which the profile entity lies is created automatically.

Opening

Creates an opening in the solid on which the profile entity lies. The opening is created by a solid on the **BC_SUBTRACT** layer. This layer is frozen by default. Thaw the layer to select the opening.

Panel type

 Single	 Double Vertical	 Double Horizontal
 Triple Vertical	 Triple Bottom	 Triple Top
 Triple Left	 Triple Right	 Triple Horizontal

7.88 BIMWINDOWUPDATE command

Updates the opening made by a window or door in a solid.

Shape Lite Pro Mechanical BIM

Icon: 

7.88.1 Description

Updates the opening created by a window or door in case the definition changed in such way that the opening was not correctly updated automatically.

7.88.2 Options within the command

MOde

Allows to select the placement face, whether Automatic or Manual.

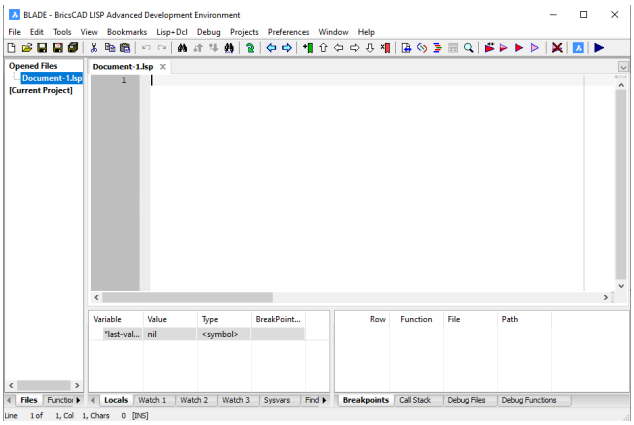
7.89 BLADE command

Opens the **BricsCAD LISP Advanced Development Environment** (BLADE).

Shape Lite Pro Mechanical BIM

7.89.1 Description

Opens the **BricsCAD LISP Advanced Development Environment** (BLADE) dialog box to interactively edit and debug LISP applications. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.



7.90 BLCOMPOSITIONS command

Opens the **Compositions** dialog box.

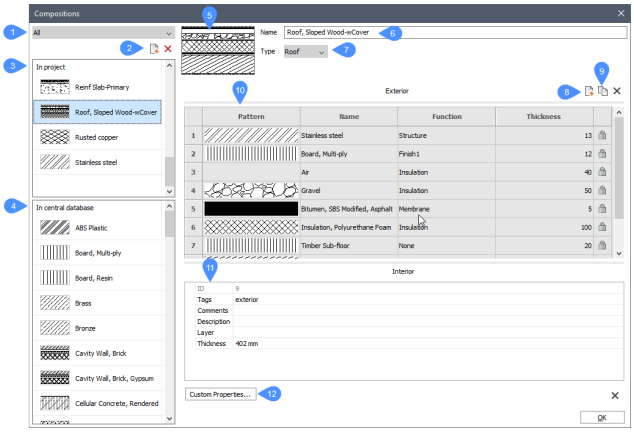
Shape Lite Pro Mechanical BIM

Icon:

7.90.1 Description

Opens the **Compositions** dialog box to create and manage BIM compositions.

The **Compositions** dialog box allows you to create, modify and delete compositions.



1 Composition category selection

2 New composition

3 In project

4 In central database

5 Preview pane

6 Name

7 Type

8 Add ply

9 Duplicate ply

10 Structure grid

11 Properties grid

12 Custom properties

7.90.2 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**.

7.90.3 New composition

Click this button to create a new composition.

7.90.4 In project

Displays the compositions of the selected category in the project database.

Right-click a composition to add a new composition, duplicate, delete or copy to the database (library).

Note: Compositions that are used in the project cannot be deleted from the project database.

7.90.5 In central database

Displays the compositions of the selected category in the central database.

Right-click a composition to add a new composition, duplicate, delete or copy to a project.

7.90.6 Preview pane

Displays the preview of the selected composition.

7.90.7 Name

Displays the selected composition name. You can change the default name.

7.90.8 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**.

7.90.9 Add ply

Opens the **Physical materials** dialog box which allows you to choose a material from the library or project database.

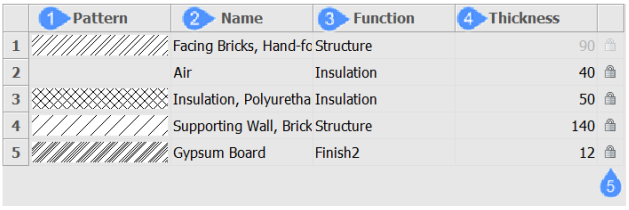
7.90.10 Duplicate ply

Inserts a copy of the selected ply.

7.90.11 Structure grid

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. Double click the **Pattern** field to change the appearance in the **Physical Materials** dialog box.

Name
Displays the name of the ply. Double click the **Name** field to change it in 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 the **Thickness** field to modify the thickness of a ply. Use the BIMUPDATETHICKNESS command to update the thickness of the solids in the model.

Note: 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.

Note: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed**: all 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.

Note: The thickness of a ply can only be unlocked if the **Variable Thickness** property in the **Physical Materials** dialog box is set to **Yes**.

7.90.12 Properties grid

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.

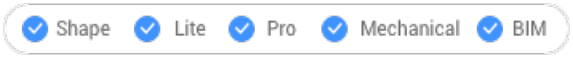
Thickness
Displays the overall thickness of a composition, which is equal to the sum of the thicknesses of the composing plies.

7.90.13 Custom properties

Opens the **Select/Create a Custom Property** dialog box. You can add custom properties for compositions.

7.91 BLIPMODE command

Toggles the BLIPMODE system variable.



Alias: BM

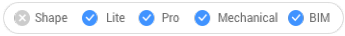
7.91.1 Description

Toggles the BLIPMODE system variable to show or hide marker blips that indicate where you picked in the drawing. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'BLIPMODE.

- On: turns on the BLIPMODE system variable
- Off: turns off the BLIPMODE system variable
- Toggle: changes the BLIPMODE system variable to the opposite of the current setting

7.92 BLMATERIALS command

Opens the **Physical Materials** dialog box.

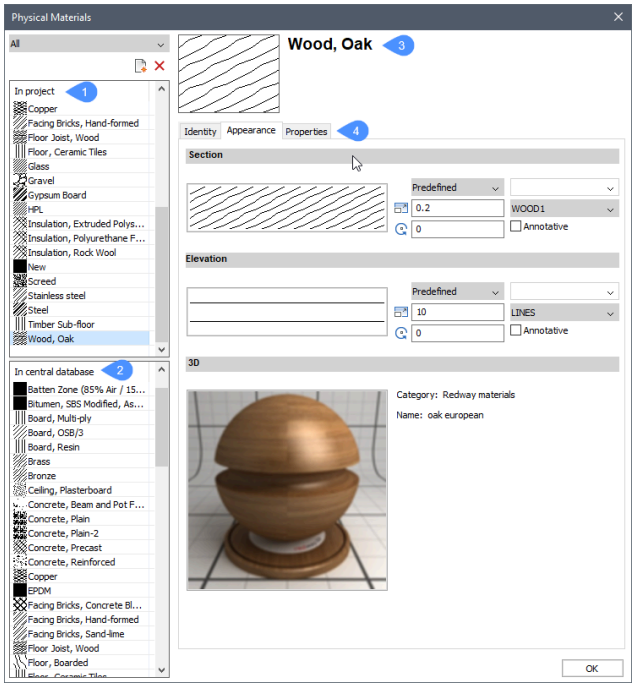


Icon: 

7.92.1 Description

Opens the **Physical Materials** dialog box to create and edit material definitions in the library or project.

The **Physical Materials** dialog box opens the Physical Materials library in the current project and the central database.



1 In project

2 In central database

3 Material name

4 Material specification tabs

7.92.2 In project

Displays all the materials that are currently loaded in the project.

7.92.3 In central database

Displays all the materials that are loaded in the central database.

7.92.4 Material name

Displays the name of the selected material.

7.92.5 Material specification tabs

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.

7.93 BLOCK command

Opens the **Create Block Definition** dialog box.

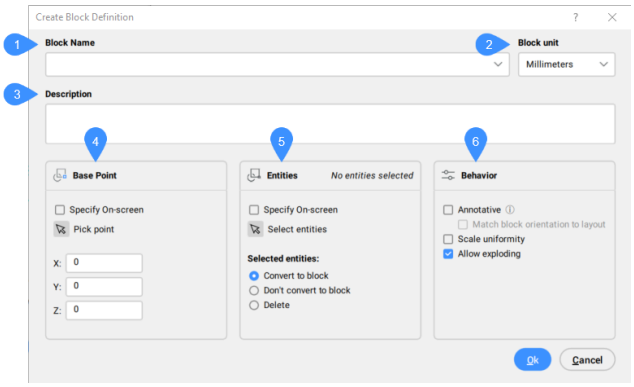
Shape Lite Pro Mechanical BIM

Icon:

Alias: B

7.93.1 Description

Opens the **Create Block Definition** dialog box to create a block definition in the current drawing.



1 Block Name

2 Block unit

3 Description

4 Base Point

5 Entities

6 Behavior

7.93.2 Block Name

Specifies the name of the block.

7.93.3 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.

7.93.4 Description

Provides a description of the block (optional).

7.93.5 Base Point

Modifies the properties of the base point of the block definition. At this point, the block is 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 ticked.

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.

7.93.6 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 **OK**, if the box is ticked.

Note: 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.

Selected entities
Defines how the entities are turned into a block.

Convert to block
Selected entities are converted to a block. This is the default option.

Don't convert to block
Selected entities are retained as entities, and the block definition is created in the drawing.

Delete
Selected entities are deleted after the block definition is created.

Note: The **Convert to block** option is the most efficient one.

7.93.7 Behavior

Annotative
Sets the annotative property of the block. This type 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.

Note: You can modify supported annotation scales of a selected block attribute in the **Properties** panel, even when its owning block reference is not annotative.

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 uniformity
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.

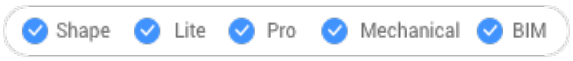
Allow exploding
Determines if users can explode the block after it is inserted. When a block is exploded, it loses its block status, and the individual entities can be edited.

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

- **On**: blocks can be exploded after being inserted, using the EXPLODE command.
- **Off**: blocks cannot be exploded. You can change this property with the EXPBLOCKS command, Blocks section.

7.94 -BLOCK command

Groups entities into a block.



Alias: -B

7.94.1 Method

Specify whether the block will follow annotative scaling and match the orientation of the viewport.

Note: Optionally, include REFERENCECURVES during the selection of entities which allows to automatically align a block during insertion.

7.94.2 Options within the command

Name for new block
Specifies the name of a new block.

?
Lists the names of existing blocks in the drawing.

Note: If an existing name is entered, BricsCAD asks you if you want to redefine the block. If it is redefined, all the instances of that block name will be changed.

Insertion point for new block
Pick a point in the drawing or enter X, Y and Z coordinates.

Note: The Z coordinate is optional, default is 0.

Annotative
Specifies if the block will follow annotative scaling.

Orient relative to sheet in paper space viewports
Determines if the annotative blocks match the orientation of the layout.

7.95 BLOCK? command (Express Tools)

Lists the entities in a block definition.

Shape Lite Pro Mechanical BIM

7.95.1 Method

Enter a block name or press Enter to select a block in the drawing, then specify an entity type. The entity list is displayed in the Command line.

7.96 BLOCKCONVERT command

Converts an AutoCAD® Dynamic Block into a BricsCAD Parametric Block.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PBLOCKASSIST, PARAMETRICBLOCKASSIST

7.96.1 Description

Converts AutoCAD® Dynamic Blocks into BricsCAD Parametric Blocks. When the command is launched, the **Block Convert** dialog box displays.

7.96.2 Method

In most common cases, the following dynamic block features are supported for conversion: visibility actions, one-grip stretch and one-grip move actions, chained actions, flip action, rotate action, array action, alignment action parameter, lookup action, with history-dependent actions, linear parameters with two grip points, one of which is not used, and block table feature.

Note: The history of the dynamic blocks will be lost during the conversion to parametric blocks. A warning message about the possible different look of converted block references will be displayed.

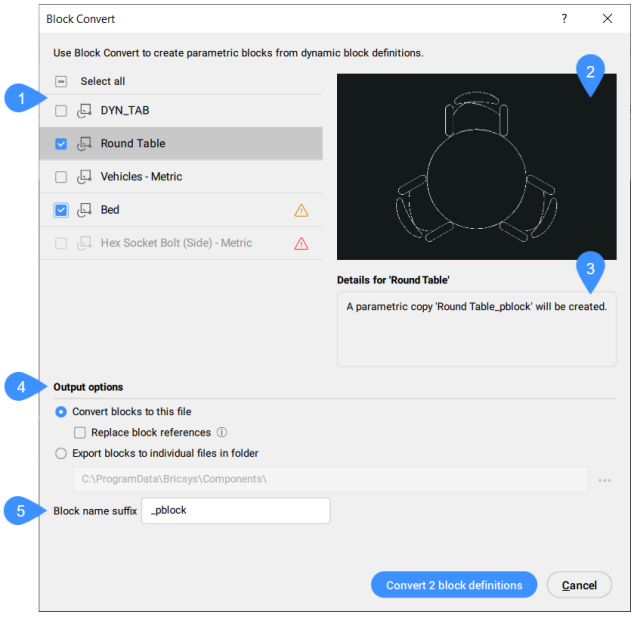
Each AutoCAD® Dynamic Block feature is reconstructed to one or several BricsCAD Parametric Block features: visibility states, parametric move operation, parametric stretch operation, linked behavior of

parametric move and stretch operations, parametric rotate operation, parametric flip operation, reference curves, design tables, parameter formulas, and associative arrays.

Once the **Block Convert** dialog box is displayed, all AutoCAD® Dynamic Blocks of the current drawing are listed. Tick all blocks to convert and specify conversion details (location, suffix).

Preselect dynamic block references by selecting the blocks before launching the BLOCKCONVERT command. The dynamic block definitions behind the selected references will be automatically selected in the **Block Convert** dialog box, and the option to convert references will be automatically toggled.

The **Block Convert** dialog box allows you to select the AutoCAD® Dynamic Blocks in the current drawing to be converted to BricsCAD Parametric Blocks.



1 List of blocks

2 Preview

3 Details for

4 Output options

5 Block name suffix

List of blocks

All AutoCAD® Dynamic blocks that are present in the current drawing are listed here. Individually check the blocks you want to convert to BricsCAD Parametric Blocks or check all by ticking the **Select all** option.

Note: Blocks that cannot be converted or can be partially converted are marked with a red or yellow warning sign respectively.

Preview

Displays a preview of the currently selected block in the list.

Details for

Displays information on whether the selected block can be converted or not. If the block cannot be converted, the features that are not supported are listed.

Some blocks can be converted with partial loss of information. In this case warning messages are shown in the details section.

Output options

Convert blocks to this file
Converts the dynamic blocks into the parametric blocks inside the current file.

Replace block references
If checked, all references to the selected dynamic blocks will be replaced with references to parametric blocks. All parameter values will be preserved.

Note: This option is only available if the **Export blocks to individual files in folder** option is not checked.

Export blocks to individual files in folder
Check this box to export the converted blocks to the specified folder. Exporting blocks to the default *C:\ProgramData\Bricsys\Components* folder makes blocks accessible through the **Library** panel. Click the **Browse...** button that opens the **Select a folder for saving exported blocks** to specify a different folder.

Block name suffix
Allows you to modify the default block name suffix. The last used name for suffix is remembered.

7.97 BLOCKICON command

Creates and updates preview bitmaps of blocks and stores them in the drawing.

Shape Lite Pro Mechanical BIM

7.97.1 Method

Type the names of the blocks to be processed. Separate block names by commas.

7.97.2 Options within the command

*

Processes all blocks in the drawing.

7.98 -BLOCKIFY command

Converts identical sets of entities (2D or 3D) to block references via Command line.

Shape Lite Pro Mechanical BIM

7.98.1 Description

Searches the drawing for identical sets of entities (2D or 3D) and replaces them with block references.

7.98.2 Method

There are four methods to use the BLOCKIFY command:

- Detect matches with selected entities
- Detect matches with existing blocks
- Detect equal 3D solids
- Detect collections (of 3D solids/block references and 2D block references)

Dependent on the selected mode, different follow-up questions and advanced options are presented.

Note: To replace block patterns by arrays of entities, use the ARRAYDETECT command. See the **ARRAYDETECT command** related article.

Select the desired method, source entities and search space. Set the options and tolerance as needed. The options displayed are relevant for the selected method and entities and reflect the BLOCKIFYMODE and BLOCKIFYTOLERANCE system variables settings (see the related articles).

The default mode is to detect matches with selected entities. This mode searches the drawing (or the selection) for recurring 2D/3D geometry and converts them to block references. Select the source entities, select the search space, select a block insertion point (for new block definitions) and specify a block name.

7.98.3 Options within the command

match Equal solids
Searches the drawing (or the selection) for equally shaped solids and converts them to block references (new block definitions). In case a solid matches an existing block definition, the replacing block reference points to that block definition.

Note: This option is not available in the BricsCAD Lite license level.

Entire drawing
Selects the entire drawing as a search space.

find Collections
Searches the drawing (or the selection) for sets of identical collections of:

- 3D solids and/or 3D block references
- 2D block references

Entire drawing
Selects the entire drawing as a search space.

Create
Replaces the collections within the selected set with block references of the new definition.

cAncel
Ends the command.

Recalculate
Recalculates the collection sets taking into account the newly created block reference (some previously found collection sets configurations might no longer be available).

Note: With a large number of input entities, calculating all possible collections can take a long time. Press the **Esc** key to cancel the calculation at any time, and then select from the collection sets found so far.

match existing Blocks
Searches the drawing (or the selection) for matches to existing block definitions present in the drawing and converts them to block references.

Entire drawing
Selects the entire drawing as a search space.

check Library
Allows you to specify the block definitions search location.

Library
Search for source blocks in the blocks library.

select Folder
Indicate the custom file/folder path to search the source blocks.

Entire drawing
Selects the entire drawing as a search space.

7.99 BLOCKIFY command

Converts identical sets of entities (2D or 3D) to block references.

Shape Lite Pro Mechanical BIM

Icon: 

7.99.1 Description

Searches the drawing for identical sets of entities (2D or 3D) and replaces them with block references.

7.99.2 Method

There are four methods to use the BLOCKIFY command:

- Detect matches with selected entities
- Detect matches with existing blocks
- Detect equal 3D solids
- Detect collections (of 3D solids/block references and 2D block references)

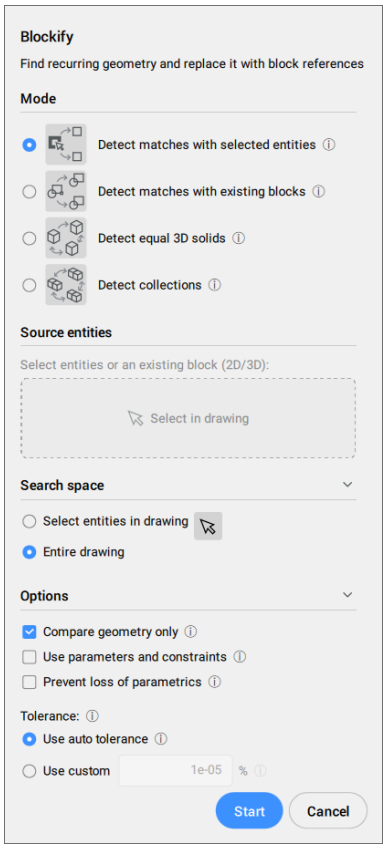
Dependent on the selected mode, different follow-up questions and advanced options are presented.

Note: To replace block patterns by arrays of entities, use the ARRAYDETECT command. See the **ARRAYDETECT command** related article.

Select the desired method, source entities and search space. Set the options and tolerance as needed. The options displayed are relevant for the selected method and entities and reflect the BLOCKIFYMODE and BLOCKIFYTOLERANCE system variables settings (see the related articles).

7.99.3 Detect matches with selected entities

Searches the drawing (or the selection) for recurring 2D/3D geometry and converts them to block references.



Source entities

Click **Select in drawing** button. Select entities or an existing block (2D/3D) in the drawing to compare other entities to.

Search space

Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing

Uses the entire drawing as search space.

Options

Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Use parameters and constraints

Try to find matching geometry by varying parameter values.

Note:

- When the option is active, a selection set containing constraints between possible matches (external constraints) is not considered valid. In this case, the Command line displays a warning prompt to remove these constraints before using the command.
- Potentially slower on large files.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

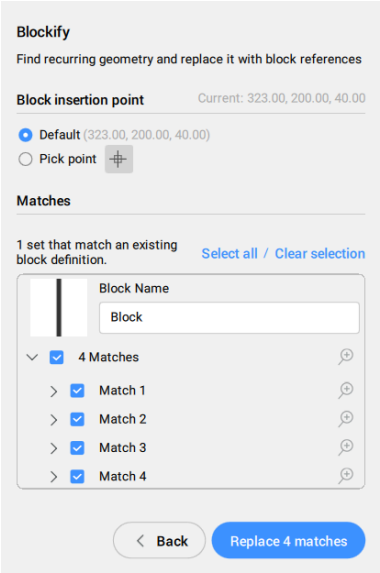
This option is highly recommended for most cases.

Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



Block insertion point

Define the insertion point of the new block definition.

Default

Accept the default insertion point marked by a red X in the model space.

Pick point
Allows you to define a new insertion point. Press the icon next to the option and select a point in the model space.

Matches
Displays a list of detected matches and allows you to select the ones to replace with block references:

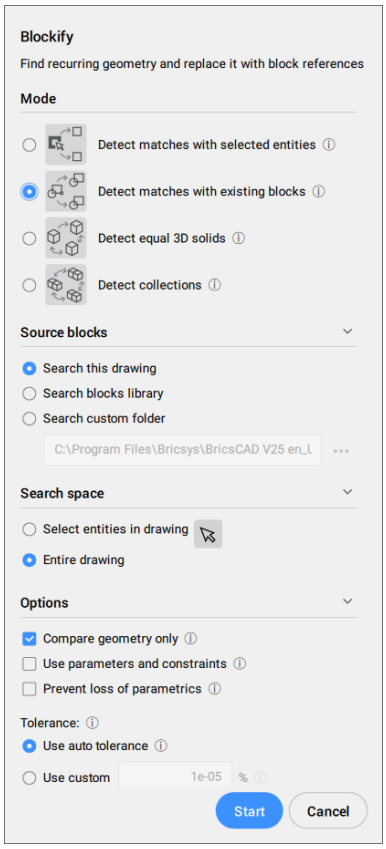
- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.
- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

7.99.4 Detect matches with existing blocks

Searches the drawing (or the selection) for matches to existing blocks from a specified location and converts them to block references.



Source blocks
Allows you to specify the source blocks search location.

Search this drawing
Search for source blocks in the current drawing.

Search blocks library
Search for source blocks in the blocks library.

Search custom file/folder
Check this radio button to activate the input field. Click the menu button to open the **Select Folder** dialog box and indicate the custom file/folder.

Search space
Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing
Uses the entire drawing as search space.

Options
Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only
Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Use parameters and constraints
Try to find matching geometry by varying parameter values.

Note:

- When the option is active, a selection set containing constraints between possible matches (external constraints) is not considered valid. In this case, the Command line displays a warning prompt to remove these constraints before using the command.
- Potentially slower on large files.

Prevent loss of parametrics
If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

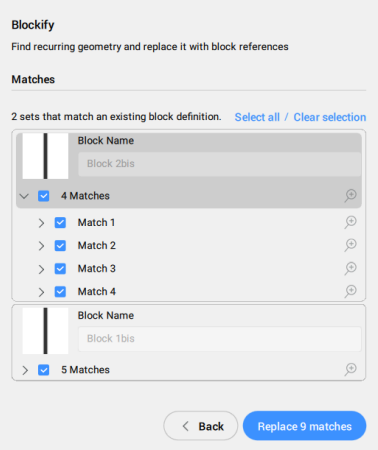
Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance
Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.
For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance
Check this radio button to use an automatically set value.
The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).
This option is highly recommended for most cases.

Use custom
Check this radio button to activate the input field.
The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



Matches

Displays a list of detected matches and allows you to select the ones to replace with block references:

- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

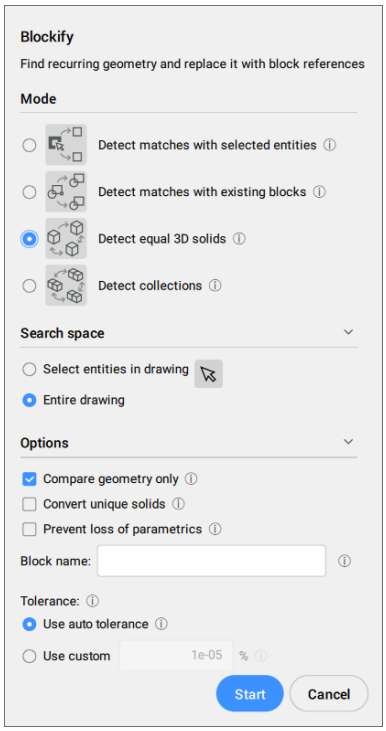
Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.
- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

7.99.5 Detect equal 3D solids

Searches the drawing (or the selection) for equally shaped solids and converts them to block references (new block definitions). In case a solid matches an existing block definition, the replacing block reference points to that block definition.

Note: This option is not available in the BricsCAD Lite license level.



Search space

Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing

Uses the entire drawing as search space.

Options

Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Convert unique solids

Tick the check-box to also convert solids to blocks when they occur only once.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Block name

Specify a block name or use the default name. If you choose to use the default name, block definitions with names like **Block1**, **Block2** etc. are created.

Note: If BLOCKIFY is used on BIM entities, it checks the BIM Name property of all equal-shaped BIM classified solids and tries to find the longest common substring to use it as the name of the block definitions.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

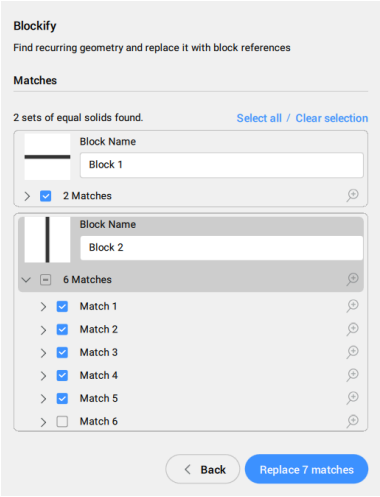
This option is highly recommended for most cases.

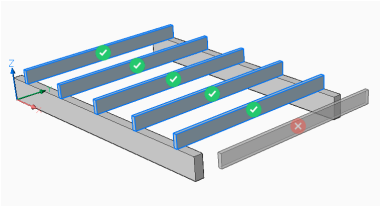
Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.





Matches

Displays a list of detected matches and allows you to select the ones to replace with block references:

- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.

- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

7.99.6 Detect collections

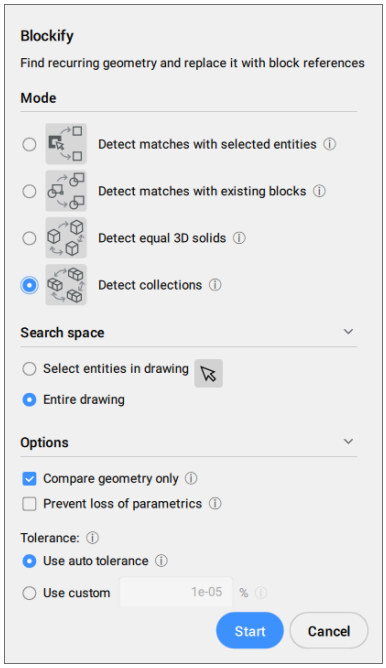
Searches the drawing (or the selection) for sets of identical collections of:

- 3D solids and/or 3D block references
- 2D block references

Allows you to select one set at a time from which to create a new block definition. Replaces the collections within the selected set with block references of the new definition.

Recalculates the collection sets taking into account the newly created block reference (some previously found collection sets configurations might no longer be available).

Note: With a large number of input entities, calculating all possible collections can take a long time. Press the **Esc** key to cancel the calculation at any time, and then select from the collection sets found so far.



Search space

Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing

Uses the entire drawing as search space.

Options

Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

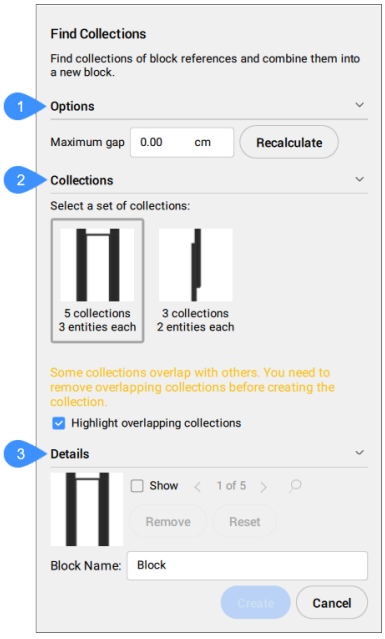
This option is highly recommended for most cases.

Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



1 Options

2 Collections

3 Details

Options

Maximum gap

Increases the allowed gap between entities to find a more complex collection.

For example: If a drawing contains multiple block references of desks and chairs, you can find a collection of both elements by setting the maximum gap between a desk and a chair.

Recalculate

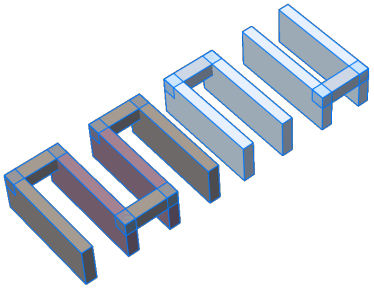
Recalculates the collections based on the new maximum gap.

Collections

Displays the found collection sets. Select the collection set you want to create the block reference from.

Note:

- A message is displayed if the selected set contains overlapping collections. Overlapping collections are collections that have at least one entity in common. Select which overlapping collections to remove so you can create the new block definition.
- The overlapping collections are highlighted if the **Highlight overlapping collections** option is ticked.



Details

Show

Tick the check-box to highlight in the **Model space** one collection at a time within the collection set.

To cycle through the collections, use the left and right arrows next to the **Show** check-box.

Use the **Zoom in** button to extend the view to the highlighted collection.

Remove

Removes the current highlighted collection from the collection set.

Reset

Resets to the initially found collections in the collection set by re-adding any collections that were removed.

7.100 BLOCKREPLACE command

Replaces block definitions.

Shape Lite Pro Mechanical BIM

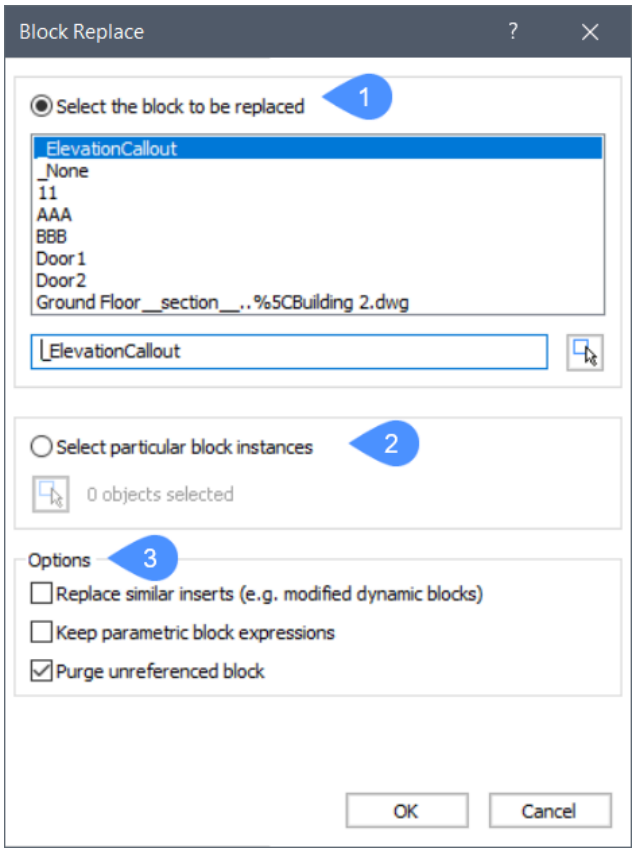
Icon:

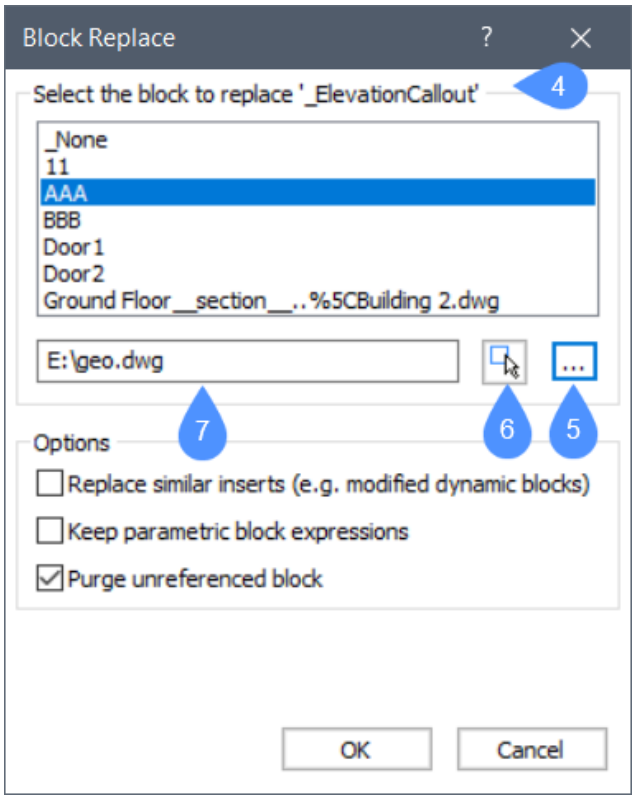
7.100.1 Method

This command displays the **Block Replace** dialog box.

Note:

- Modified (anonymous) instances of dynamic blocks cannot be selected by name.
- Externally dependent blocks cannot be replaced.
- When a block with attributes is replaced by a block without attributes, the replaced block reference keeps the old attributes.
- When a block without attributes is replaced by a block with attributes, the replaced block references doesn't insert new attributes.
- When an annotative block is replaced with a non-annotative block, the block reference keeps the old context data manager and annotation scales in the extension dictionary.
- When a non-annotative block is replaced with an annotative block, BricsCAD adds context data immediately according to CANNOSCALE.
- When a dynamic block is replaced by a non-dynamic block or another dynamic block, the old extension dictionary **AcDbBlockRepresentation** is removed from the block reference.
- Dynamic blocks that are replaced keeps their parameter values if the replacement blocks have parameters with the same name and type.





1 Select the block to be replaced.

2 Select particular block instances.

3 Options.

4 Select the block to replace selected block instances.

5 Choose a .dwg or a .dxf file instead of a block name.

6 Select a block by clicking on it.

7 The blockName.

7.100.2 Select the block to be replaced

Selects a block by name to be replaced.

Note: The modified (anonymous) dynamic blocks cannot be selected by name.

7.100.3 Select particular block instances

Selects any block instances, including modified (anonymous) dynamic block instances to be replaced.

7.100.4 Options

Replace similar inserts

Replaces all modified (anonymous) and unmodified instances of dynamic block and all instances of non-dynamic block selected.

Keep parametric block expressions
Keeps the parameters' expressions of the old block if the replacement block has parameters with identical names.

Purge unreferenced block
Removes unreferenced block instances.

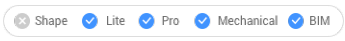
Select the block to replace selected block instances by

- Choosing a block name from the list.
- (5) Browsing for a .dwg or a .dxf file (the file name will be the block name).
- (6) Selecting a block by clicking on it.
- (7) Typing a file name of an external .dwg or .dxf that will be used as a new block name (e.g. fileName.dwg) or typing "blockName=fileName.dwg" when the new block name should be different than the file name.
- ***Note:*** fileName.dwg can have blocks with the same names as in the current drawing. In this case, blocks from the file do not re-define blocks with the same names already present in the current drawing.

  Note: If fileName.dwg file is not located in the support file search path, you have to indicate the complete path.

7.101 -BLOCKREPLACE command

Replaces block definitions.



7.101.1 Method

There are two methods to replace block definitions:

- By replacing all block instances.
- By replacing particular block instances.

Note:

- Modified (anonymous) instances of dynamic blocks cannot be selected by name.
- Externally dependent blocks cannot be replaced.
- When a block with attributes is replaced by a block without attributes, the replaced block reference retains the old attributes.
- When a block without attributes is replaced by a block with attributes, the replaced block references doesn't insert new attributes.
- When an annotative block is replaced with a non-annotative block, the block reference keeps the old context data manager and annotation scales in the extension dictionary.
- When a non-annotative block is replaced with an annotative block, BricsCAD adds context data immediately according to CANNOSCALE.
- When a dynamic block is replaced by a non-dynamic block or another dynamic block, the old extension dictionary **AcDbBlockRepresentation** is removed from the block reference.

7.101.2 Options within the command

Enter name of block to be replaced
Enter the name of the block to be replaced. Note that modified (anonymous) instances of dynamic blocks cannot be selected by name.

?
Lists the block names.

select object (=)
Select a block reference with the desired name. Note that modified (anonymous) dynamic blocks cannot be selected.

options (*)
Lists the replace options.

by block Name
Enter a block name to replace. Note that modified (anonymous) dynamic blocks cannot be selected by name.

Selected block instances
Selects any block instances, including modified (anonymous) dynamic block instances to be replaced.

SIMilar inserts
Replaces all modified (anonymous) and unmodified instances of dynamic blocks and all instances of non-dynamic blocks selected.

Enter block name or file to replace selected entities
Enter a block name or file to replace selected entities as follow:

- blockname: name of a block already defined in the current drawing;
- fileName.dwg: name of an external .dwg or .dxf file (file name will be used as a new block name);
- blockName=fileName.dwg: when new block name should be different than fileName.

Note: fileName.dwg can have blocks with the same names as in the current drawing. In this case blocks from file do not re-define blocks with the same names already present in the current drawing.

Note: The file name should include the entire path if the file is not located in the support file search path.

Enter a block name or a file name to replace selected entities. The file name should include the entire path if the file is not located in the support file search path.

Purge unreferenced when finished?
Removes unreferenced block instances.

7.102 BLOCKTOXREF command

Replaces all instances of a block with an Xref.

Shape Lite Pro Mechanical BIM

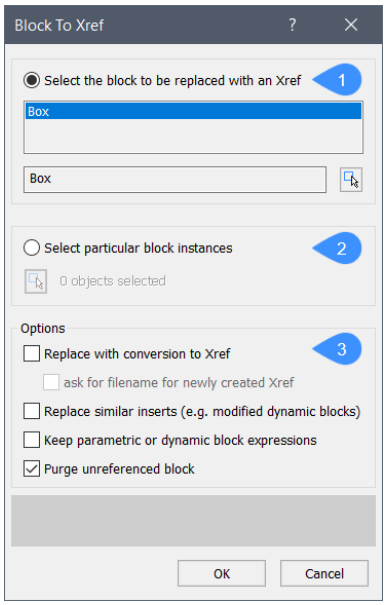
Icon: 

7.102.1 Method

Select one or more block references to replace with Xref/s.

This command displays the **Block To Xref** dialog box.

Note: The block name for the Xref is generated from the file name. If an Xref (or overlay) with the same name already exists in the drawing, the block is replaced using existing Xref.



1 Select the block to be replaced.

2 Select block instances.

3 Options.

7.102.2 Select the block to be replaced

Selects all block instances to replace.

7.102.3 Select particular block instances

Selects only particular block instances to replace.

7.102.4 Options

Replace with conversion to Xref
ON: saves the block first and then replaces the block with the newly created Xref.
OFF: the block is replaced with an Xref.

ask for filename for newly created Xref
ON: the **Save Block** dialog box displays to save the block before replacing it with an Xref.

Note: If the selected block name is a dynamic block name and the **Replace similar inserts** option is On, all modified dynamic blocks are replaced by one Xref. If the selected block is an instance of a modified dynamic block and the **Replace with conversion to Xref** option is On, the newly created Xref will contain the default (unmodified) dynamic block definition (similar with WBLOCK command which cannot save anonymous block definitions).

OFF: a new drawing is created next to the original drawing with the same name as the block name.

Replace similar inserts (e.g. modified dynamic blocks)

Note: This option acts like the BLOCKREPLACE command.

ON: replaces all instances of a block and all its modified (anonymous) dynamic blocks.

Note:

- If the **Select particular block instances** is On and a non-dynamic block is selected, all instances with the same block name are replaced with the Xref.
- This option cannot be changed when the **Replace with conversion to Xref** option is also On and **Select particular block instances** is selected, because multiple Xref files and multiple block definitions can be created for one block.

OFF: anonymous (modified) dynamic blocks cannot be replaced with Xref.

Note: If **Select particular block instances** is selected, only selected instances are replaced with Xref.

Keep parametric or dynamic block expressions
Keeps expressions of the replaced block when the replacement block has parameters with the same names.

Purge unreferenced block
Removes unreferenced block instances.

7.103 -BLOCKTOXREF command

Replaces all instances of a block with an Xref at the command line.

Shape Lite Pro Mechanical BIM

7.103.1 Method

Select one or more block references to replace with Xref/s.

Note: The block name for the Xref is generated from the file name. If an Xref (or overlay) with the same name already exists in the drawing, the block is replaced using existing Xref.

7.103.2 Options within the command

Enter block name of the block to be replaced with an xref
Enter a block name.

?
Lists the block names.

select objects (=)
Select a block reference with the desired block name.

options (*)
Lists the replacement options.

by block Name
Enter a block name to replace.

Selected block instances
Select the block instance(s) to replace.

SIMilar inserts
Replaces all instances of a block and all its modified (anonymous) dynamic blocks.

Note: If non-dynamic block is selected, then all instances with the same block name is replaced with Xref.

Note: This option acts like the BLOCKREPLACE command.

with Conversion to xref

Yes
First saves the block and then replaces the block with the newly created Xref.

No
The block is replaced with an Xref.

Ask for filename for newly created Xref

Yes
Saves the block before replacing it with an Xref.

No
A new drawing is created next to the original drawing with the same name as the block name.

Enter an xref file (~ to open file dialog)
Enter the name of an xref file or enter ~ to open the Select an Xref file dialog box.

7.104 BMANIMATE command

Runs a quick animation of an exploded view and exploded view steps.

Shape Lite Pro Mechanical BIM

7.104.1 Method

1 Open for edit the exploded representation using the BEDIT command.

2 Launch the BMANIMATE command.

3 Select **Exploded view** and define the first and the last steps for animation.

Note: The command runs according to the values set with the BMEXPLODESTEPEDIT command.

4 After finishing the animation, launch the BCLOSE command, to close the block editing session.

7.104.2 Options within the command

paUse
Pauses the animation.

Play
Plays the animation.

Stop
Stops the animation.

pRevious
Shows the previous step of the animation.

Show
Shows the next step of the animation.

Finish
Finishes the animation at that moment.

7.105 BMARROW command

Creates either a linear or circular arrow.

Shape Lite Pro Mechanical BIM

Icon: ↑, ↻

7.105.1 Description

Creates either a linear or circular arrow.

- A linear arrow is created between two selected points.
- A circular arrow can be created for a selected circular edge or a face with circular edges.

These linear and circular annotation arrows can be placed on 3D assemblies, indicating a part's movement to create assembly instructions. The annotations are exportable along with the geometry to the 2D format SVG to be used in instruction manuals.

7.105.2 Method

Pick the start point, the end point, and the rotation angle of the arrow.

7.105.3 Options within the command

Linear arrow
Creates a linear arrow between two selected points.

Note: This is the default option.

Circular arrow
Creates a circular arrow for a selected circular edge or a face with circular edges.

head LEngth
Sets the head length of the arrow.

head Width
Sets the head width of the arrow.

Thickness
Sets the thickness of the arrow.

7.106 BMASSEMBLYINSPECT command

Evaluates the capability of a mechanical assembly to be disassembled or assembled without collisions by respecting a given rule.

Shape Lite Pro Mechanical BIM

Icon:

7.106.1 Description

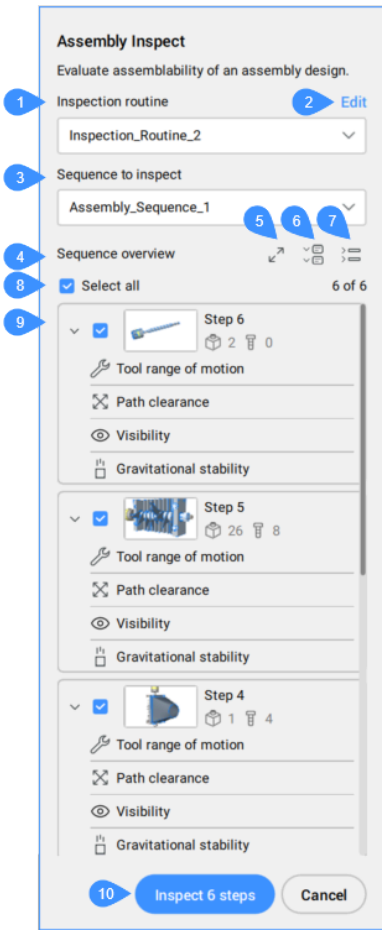
Evaluates criteria on a given assembly sequence to assess the ability to assemble your design.

Note: When launching the BMASSEMBLYINSPECT command, the User Coordinate System is used for all calculations during the entire lifetime of the command. This means that there is no impact on the results when turning on/off the UCS during the command.

7.106.2 Method

Choose whether to use the panel to specify the sequence or to define it in the Command line.

7.106.3 Assembly Inspect command context panel



1 Inspection routine

2 Edit

3 Sequence to inspect

4 Sequence overview

5 Thumbnail toggle button

6 Expanded steps display

7 Collapsed steps display

8 Select all

9 List of steps in the selected sequence

10 Inspect x steps

Inspection routine
Displays the current routine selected from the drop-down list.
If no routine is available, open the **Manage routines** from the drop-down list to create one.

Edit
Opens the **Routine editor** dialog box to edit the selected routine.

Sequence to inspect
Allows you to select a sequence to inspect from the drop-down list.

Sequence overview
Displays all steps of the selected sequence.

Note: You can select steps one by one or all together (**Select all**).

Select all
Selects or deselects all steps from the sequence at once.

Thumbnail toggle button
Switches between small and large thumbnails.
Small thumbnails give you a quick and compact overview of the sequence and assembly results, while large thumbnails allow you to navigate through the sequence and make it easier to find a specific step for activation and inspection.

Expanded steps display
Displays all steps in details.

Collapsed steps display
Collapses the displaying steps in details.

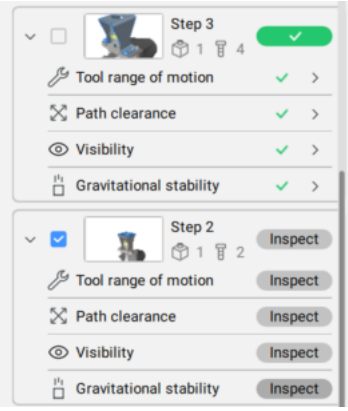
List of steps in the current sequence
Displays all steps of the current sequence.

Each step can be selected individually, has a graphic representation (thumbnail), and a list with the criteria for evaluation. Also, the total number of parts and fasteners are shown for each step.

The thumbnails and actual active space during BMASSEMBLYINSPECT respect the orientation of the step in the inspected sequence.

Criteria are applied to the assembly by respecting the step orientation (calculation, model space visualization, and temporary graphics).

By pressing the **Inspect** button, you can run the evaluation for an entire step or for each criterion from the used routine.



Inspect x steps

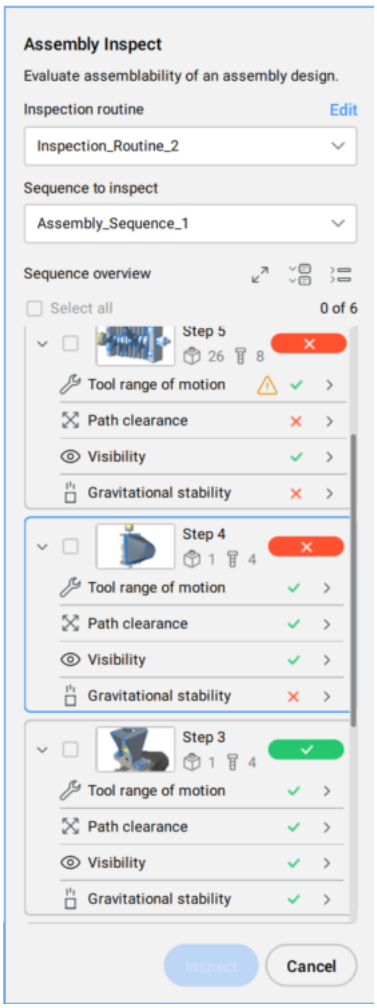
Runs the criteria included in the routine for evaluating all selected steps.

Note: You can **Stop** the evaluation process any time.

The result of the inspection provides information about how was evaluated each step.

A warning symbol is shown next to a specific criterion in a specific step in case of:

- No fasteners in the **Tool range of motion** criterion.
- No tool-fastener mappings in the **Tool range of motion** criterion.
- No parts in the **Path clearance** criterion.
- No parts in the **Visibility** criterion.
- No parts in the **Gravitational stability** criterion when the "ignore fasteners" option is enabled.

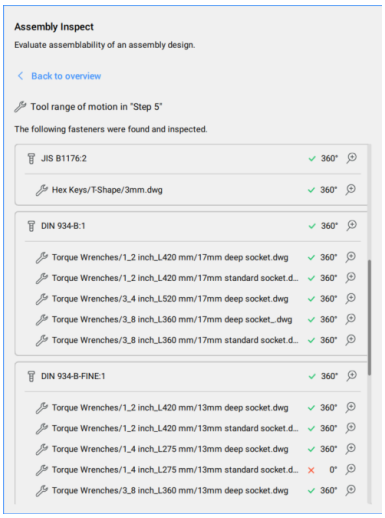


For each criterion from the routine, there is a graphic representation of the result of the evaluation. Depending on the acceptance range set for each criterion, the result of the evaluation is displayed in corresponding colors.

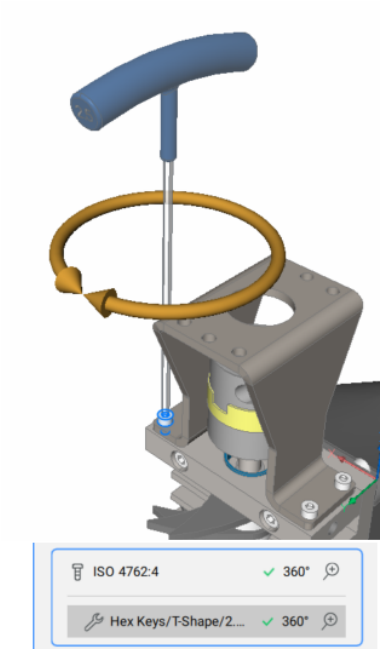
Press the right-side arrow for detailed results. Some evaluations have passed, and others have errors. You can see easily where the error occurs by pressing the Zoom button.

Example of the result of the **Tool range of motion** evaluation:

In the detailed results panel of the **Tool range of motion** criterion, each applicable tool is listed with its angular range for each listed fastener. The largest angular range of all tools applicable to this fastener defines the angular area of a fastener.



Note: After pressing the Zoom button, the tool is visualized in the drawing area by its start and end position of rotation, and an arrow in between these positions that indicate the range of motion.



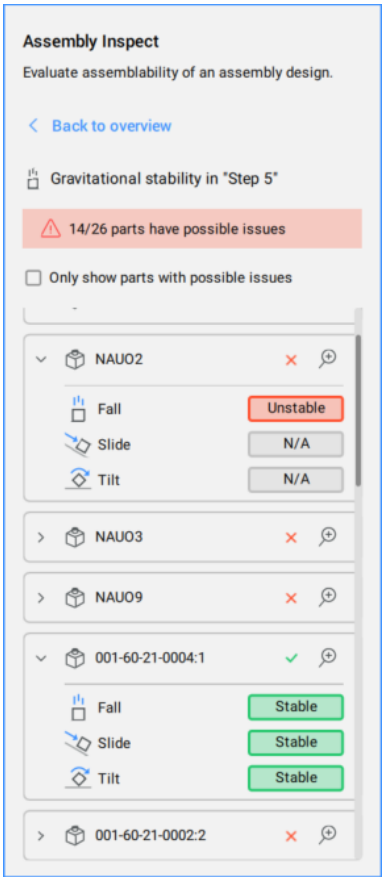
Example of the result of the **Path clearance** evaluation:



Note: Press the Zoom button to visualize the clearance directional arrow in the drawing area.

Example of the result of the **Gravitational stability** evaluation:

Note: Partial results for the three checks are shown, as well as temporary arrows in model space.



7.106.4 Options within the command

You can run the assembly inspect by selecting an option from the Command line.

Run current routine
Runs the current routine.

Cancel
Cancels the running the routine.

Exit
Exits the command.

7.107 BMBALLOON command

Creates associative balloon entities for assembly components.

Shape | Lite | Pro | Mechanical | BIM

Icon:

7.107.1 Description

Creates associative balloon entities for mechanical and non-mechanical components in the Model Space and for their generated views in a Paper Space layout. Each balloon contains an index of the mechanical component, or an id string for a non-mechanical solid or a block in the associated BOM table created by the BMBOM command.

Note: In the **BOM Manager** panel set the **Column role** option as **Number** for the id string for a non-mechanical solid or a block which will be displayed in the balloon.

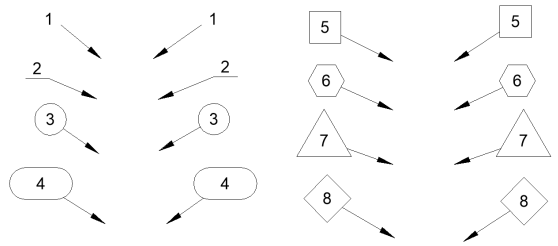
Note: Objects with non-empty id that are not found in BOM tables will be marked as invalid and will display "?" in the balloon.

Note: Balloons are created as Multileader entities using the current Multileader style. Create a Multileader style with the **Content** set to **Block** and adjust the **Scale** property to control the size of the balloon frame.

7.107.2 Options within the command

select other Table
Allows you to select a different BOM table in case multiple tables exist in the layout. The most recently inserted BOM table is selected by default.

select balloon Style
Allows you to define the shape of the balloon.



Note: Predefined balloon styles from _BalloonStyles.dwg can be used, as well as any custom user-defined styles.

Note: The PROMPTMENU variable allows the selection of the balloon styles.

Auto mode
Allows automatic placement of balloons only on components listed in the specified BOM table in a particular drawing view.

placement Mode
Specifies whether multiple balloons with the same values are generated or not.

Unique
Only unique values are generated on one view.

All
All values are generated on one view.

Number mode

Consecutive
Enumerates the balloons in consecutive order during a circle or a polyline pattern.

Circle
Enumerates the balloons in consecutive order to a circular pattern.

Toggle direction
Toggles the numeration direction between clockwise and counterclockwise. Same action occurs by pressing Ctrl key.

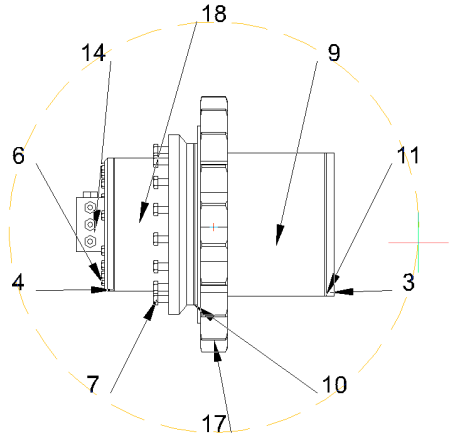
move start Forward
Moves the start balloon forward. Same action occurs by pressing Tab key.

move start Backward
Moves the start balloon backward. Same action occurs by pressing Shift+Tab key.

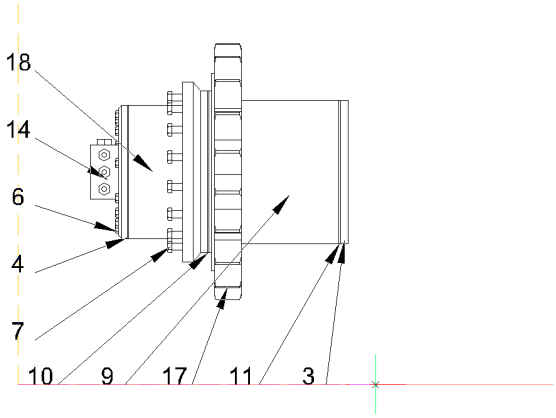
by BOM table
Places the balloons with respect of the BOM table order.

Note: The BOM table will be reordered accordingly to the balloons order.

Circle
Aligns leaders along a circle.



Polyline
Aligns leaders along a polyline.



Back
Returns to previous prompt.

7.108 BMBOM command

Inserts a Bill of Materials (BOM) table.

Shape | Lite | Pro | Mechanical | BIM

Icon:

7.108.1 Description

Inserts a Bill of Materials (BOM) table in the current drawing.

The BOMFILTERSETTINGS system variable controls which components will be included in the BOM table.

7.108.2 Options within the command

Name
Specifies the table name.

Note:

- The BOMTEMPLATE system variable defines the path to the file with default BOM template which will be used to create new BOM tables.
- If the BOMTEMPLATE system variable is empty or the path is not valid, the below default BOM configuration will be used.

Top level
Creates a BOM of the top-level components only.

Bottom level
Creates a BOM of the sub-components.

Hierarchical
Creates a hierarchical BOM listing all (sub)components.

Load from template
Opens the **Select file with the BOM table template** dialog box which allows you to select a BOM file to create a BOM from a template.

Save as template
Opens the **Select file to save BOM table template** dialog box which allows you to save the current BOM configuration as a template.

LAyout
Changes the layout where the table will be placed.

Configure
Allows you to add additional columns to your BOM table.

Description
Adds a description column.

dEnsity
Adds a density column.

Volume
Adds a volume column.

Mass
Adds a mass column.

parameterS
If multiple inserts of the same parametric component exist in the assembly, they will be grouped in different BOM rows depending on the values of the parameters.

mAterials
Adds a material column.

Thickness
Adds a thickness column.

Note: If a component is a sheet metal part, this column shows the thickness of the part. Otherwise, it is empty.

MOre
Displays more configuration options.

Toggle column
Adds or removes columns from the BOM.

add Formula field
Adds a formula field.

Note: Use < > to refer to other columns.

add Thumbnails
Adds a thumbnail column.

Name
Sets the name of the column.

Width
Sets the thumbnails width in pixels.

Height
Sets the thumbnails height in pixels.

Block Scale
Sets the block scale for thumbnails.

View type
Sets the orientation of the camera.

Orthographic
Allows you to choose an orthographic projection view.

Isometric
Allows you to choose an isometric projection view.

viSual style
Sets the visual style for thumbnails.

BackGround color
Sets the thumbnails background color.

Transparent
Sets transparent background for thumbnails.

seleCt color
Opens the **Color** dialog box to select a color.

Folder
Opens the **Select image folder** dialog box which allows you to choose a folder for thumbnail images.

Add column
Adds a thumbnail column with specified parameters to the BOM table.

Set sorting
Sets the sorting mode for the BOM.

Automatic
Sets the automatic sorting for the table.

No sorting
Disables the sorting for the BOM.

Custom sorting
Specifies the column for sorting.

Toggle sorting column
Sets the number of the ordering column for sorting.

Fixed order
Keeps the initial numbers position of parts in the BOM, regardless of added or deleted parts.

column Properties
Configures columns properties. It depends on the column type.

Name
Sets a name for the column.

Visibility
Sets the visibility of the column.

ON
Makes the column visible.

OFF
Makes the column invisible.

Width
Sets the column width.

Auto
Sets the automatic width of the column.

Thumbnails Parameters
Sets the thumbnails parameters for the added thumbnails column.

Footer type
Sets the footer type.

None
Removes the column from the footer.

Sum
Adds the total sum to the footer.

Average
Adds the average value to the footer.

MINimum
Adds the minimal value to the footer.

MAXimum
Adds the maximal value to the footer.

Note: It is possible to add footers to formula or template columns as well.

Note: A different unit and format for footer values can be set.

Prefix
Sets the prefix for numbers in the **Number** (No.) column.

Suffix
Sets the suffix for numbers in the **Number** (No.) column.

Delimiter
Sets the delimiter between levels in numbers for hierarchical BOM tables.

NumeraTion type
Sets the numeration type for hierarchical BOM tables.

Full numbers
Numbers for each level are combined using the provided delimiter and shown in the **Number** column.

Short numbers
Only numbers for the current level of each part are used. This mode is supposed to be used with **Level** column.

Continuous numbers
All parts are numbered continuously, regardless of their level.

Aggregate function
Sets the aggregate function.

None
Removes an aggregate function from the column.

COUNTs
Defines aggregate function.

TOtal count
Shows the total number of encounters with the same value among the groups.

AVerage count
Shows the average number of encounters with the same value among the groups.

MINimum count
Shows the minimum number of encounters with the same value among the groups.

MAXimum count
Shows the maximum number of encounters with the same value among the groups.

LEast frequent
Shows the value with least corresponding counts. If there are more than one value with the minimal count, then all those values are displayed.

MOst frequent
Shows the value with most corresponding counts. If there are more than one value with the maximal count, then all those values are displayed.

CONcatenate
Combines all values in the group using the provided delimiter.

CONCatenate with count
Combines all values in the group with their number using the provided delimiters, prefix and suffix.

sEttings
Configures the aggregated column.

value Separator
Sets the delimiter between concatenated values.

count Position
Sets the position of the number of encounters for concatenated values.

After value
The number of instances is placed after the corresponding value.

bEfore value
The number of instances is placed before the corresponding value.

Count separator
Sets the delimiter between a value and its number of encounters.

count PREfix
Sets the prefix that will be added before the number of encounters.

count SUFfix
Sets the suffix that will be added after the number of encounters.

Units
Configures units and how they display.

unit Mode
Sets the units for the column values.

Best for All
Sets the best unit for all values.

BEst for each value
Sets the best unit for each value.

set Fixed unit
Allows you to choose the unit manually.

unit Format
Sets how the units display.

Title
Places the unit symbol in the column title.

Same cell
Places the unit symbol in the same cell as the value.

separate Column
Places the unit symbol in a separate column.

do Not show units
Hides the units symbol.

title Format
Sets the format for the column title for the Title option.

toggle excluded parameTErs
Sets the parameters to be excluded.

fOrmat string
Sets the format string for column values.

Footer format
Sets a footer format string for footer values. If the footer format is not set, then the **Column format** string will be used.

Role
Sets the column role for the column.

Regular
Sets the regular role for the column.

Number
Sets the number role for the column. (So it will be used as the number source for the BMBALLOON command).

nAme
Sets the name role for the column. (So it will be used as the name source for the BMBALLOON command).

Quantity
Sets the quantity role for the column. (So it will be used as the quantity source for the BMBALLOON command).

table sEttings
Configures the properties of a BOM table.

footer Title
Sets the title of the footer row.

Filter
Sets the table filter.

Note: The BOMFILTERSETTINGS system variable sets the default filter settings to define which objects to include.

change Mode
Sets the mode to include or exclude objects.

Components / ExClude components
Includes / Excludes mechanical components, block and solids.

Xrefs / EXclude xrefs
Includes / Excludes non-mechanical external references as top level objects.

Xrefs as Transparent
Treats external references as transparent.

Others / Exclude Others
Includes / Excludes non-mechanical local block and solids.

Plies / Exclude Plies
Includes / Excludes solid plies.

Ignore bom status / Respect bom status
Ignores / Respects BOM status of existing objects.

Stop at Leaf Parts / Transparent Leaf Parts
Excludes/includes non-mechanical objects inside components to be extracted to the BOM table.

Note: To switch the flag, type accordingly **TLP** or **SLP** in the Command line.

Property set
Specify the properties to add from.

Note: The BOMPROPERTYSET system variable sets the default set of properties for BOM tables.

Mechanical only
Only mechanical properties of components and instances will be available.

all Except coordinates
All properties of mechanical components and instances will be available, including properties of associated database entities, except coordinates.

All
All properties of mechanical components and instances will be available, including properties of associated database entities.

Grouping mode
Specifies the grouping mode for the current table.

Auto
Groups the parts with respect to their definitions and parameters.

by COmponents and columns
Groups the parts with respect to their definitions and properties used in the table.

by Columns only
Groups the parts with respect to their properties used in the table only. (So one row may correspond to completely different parts if their properties displayed in BOM are the same).

Counting mode
Specifies the counting mode for hierarchical BOM. Available for Hierarchical mode.

by Document
Sets the counting mode to count all instances.

by parent Component
Sets the counting mode to count instances in parent component.

max Level
Allows you to specify the maximal level of parts in hierarchical BOM tables.

Selection set
Allows you to specify a selection set linked to this BOM table.

Entire model
Sets entire model as the selection set.

Subassembly
Sets a selected subassembly as the selection set.

Custom selection
Sets a custom selection of top-level components as the selection set.

drawing View
Associates this BOM table with selected drawing views.

layout viewPOrt
Associates this BOM table with selected layout viewports.

eXclude current document
Excludes the currents document.

Other documents
Includes parts from other documents.

Add drawing or sheet set
Adds drawings or sheet sets from the **Select drawing file or sheet set** dialog box that is opened.

add Folder
Adds an entire folder selected from the **Folder Options** dialog box. The user may include only the documents that match specified wildcards, entire BIM project or entire sheet set.

> ***Note:*** Internally, BOM uses the logic implemented in the data extractor. It means that it will check for duplicated top-level objects, but if one document refers to another via an external reference, and that another document is also explicitly added, then the content of that document will be counted twice. When the document is saved, all paths are converted to be relative to that document when possible.

fixed coRner
Changes the fixed corner of the table. The fixed corner will remain in position during table changes.

> ***Note:*** Press the **Tab** key to cycle the corners of the table. The table position will change to place the fixed corner under the cursor, thus helping to align the table with other objects in the drawing.

Top Left
Sets top left corner of the table as fixed.

Top Right
Sets top right corner of the table as fixed.

Bottom Left
Sets bottom left corner of the table as fixed.

Bottom Right
Sets bottom right corner of the table as fixed.

Back
Returns to the previous prompt.

7.109 BMBOMEDIT command

Edits an existing Bill of Materials (BOM) table in the current drawing.

Shape Lite Pro Mechanical BIM

7.109.1 Options within the command

List tables
Lists all BOM tables in the current drawing, from which you can select one to edit.

Note: Also, you can edit the BOM tables that only exist in the **BOM Manager** panel.

Current bom
Edits the current BOM table.

Erase
Removes the selected BOM table.

Update all
Updates all BOM tables.

Erase all
Removes all BOM tables.

APPLY
Applies the current changes.

DISCARD
Discards the current changes.

Note: See the BMBOM command for the full command options.

7.110 -BMBOMEXPORT command

Exports the content of a BOM table as a table with link, or to a CSV or XLSX file.

Shape Lite Pro Mechanical BIM

7.110.1 Options within the command

Options for source BOM table

Bom table
Defines the BOM table that will be exported.

List tables
Lists the available BOM tables in the current document.

Current bom
Selects the current BOM.

template File
Defines the BOM table template that will be exported. The file path name has to be specified in the Command line.

Options for target

Table with link
Exports the content of the BOM table to another table with datalink. Opens the **Select file to save the BOM table template** dialog box.

csv or excel File
Exports a BOM table to a CSV or XLSX file. The file path name has to be specified in the Command line.

7.111 BMBOMEXPORT command

Exports the content of a BOM table as a table with link, or to a .csv or .xlsx file.

Shape Lite Pro Mechanical BIM

7.111.1 Options within the command

Options for source BOM table

Bom table
Defines the BOM table that will be exported.

List tables
Lists the available BOM tables in the current document.

Current bom
Selects the current BOM.

template File
Defines the BOM table template that will be exported. Opens the **Select file with a BOM table template** dialog box.

Options for target

Table with link
Exports the content of the BOM table to another table with datalink. Opens the **Select file to save the BOM table template** dialog box.

csv or excel File
Exports a BOM table to a CSV or XLSX file. Opens the **Select file to export BOM table** dialog box.

7.112 BMBOMPANELCLOSE command

Closes the **BOM Manager** panel.

Shape Lite Pro Mechanical BIM

7.112.1 Description

Closes the **BOM Manager** panel to hide it from the current workspace. If the **BOM Manager** panel is stacked when you close it, its the BOM manager tab or icon is removed from the stack.

7.113 BMBOMPANELOPEN command

Opens the **BOM Manager** panel.

Shape Lite Pro Mechanical BIM

7.113.1 Description

Opens the **BOM Manager** panel to display it in the current workspace. The **BOM Manager** panel appears at the same size and location as it did before it was closed or collapsed. Like any other dockable panel, the **BOM manager** panel can be either floating, docked or stacked.

7.114 BMBOMTEMPLATEEDIT command

Creates or edits a standalone Bill of Materials (BOM) template files using both the command line and the **BOM Manager** panel.

Shape Lite Pro Mechanical BIM

7.114.1 Description

Creates or edits a BOM template file using both BMBOM command and the **BOM Manager** panel. For creating a new .bom template file, the user has to provide a non-existing file name. After the .bom file is selected, the **BOM Manager** panel will be opened.

The **BOM Manager** will show a BOM table created using the selected file with the current document as the source of objects. In that mode, the **BOM Manager** will work mostly similarly to the normal mode, with following key differences:

- Instead of the list of BOM tables in the document, the top selector will show the name of the current template only.
- Commands to create a new BOM table are disabled.
- Refreshing the content of the BOM Manager button is disabled.
- All selection set modes except "Entire model", "Exclude current document", and "Additional documents" will be disabled. The property to set a target component will not be present as well.
- If the button to place a BOM table is pressed, then the command will switch to the table placement mode. After the table is placed in the document, the command will finish as if "OK" button was pressed.
- This command will not affect existing BOM tables in the document, nor will add any new BOM table to the document. The "Place BOM" command simply will create a table with BOM content, but this table will be not associated with any BOM table in the document.

7.114.2 Options within the command

Supplementary options in the command line, other than for the BMBOM command

APPLY
Applies the current changes.

DISCARD
Discards the current changes.

Note: See the BMBOM command for the full command options.

Supplementary options in BOM Manager panel

OK
The template .bom file will be updated according to the configuration in the BOM Manager, and the command will finish.

Cancel
All changes to the template .bom file will be discarded.

Note: In both cases, the BOM Manager will switch back to show the current BOM table in the document. If the BOM Manager has not been active at the start of the command (i.e, not visible and not stacked), the command will close it automatically.

Note: See the **BOM Manager** panel

Note: The BMBOMTEMPLATEEDIT will always use filter settings, grouping mode and property set suitable for BIM models, even if no BIM project is found.

7.115 BMCONNECT command

Connects two components to each other.

Shape Lite Pro Mechanical BIM

Icon:

7.115.1 Description

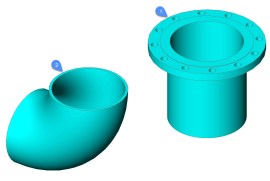
Connects two components by creation of 3D constraints between their connection entities. Also, under the **complete flange Assembly** option, a gasket and a bolting assembly are inserted, resized, and connected to a pair of flanges.

7.115.2 Method

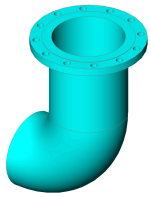
Select components to connect
Select the component to connect. (1)

Select target component
Select the component to connect to. (2)

Note: Mechanical blocks are also accepted.



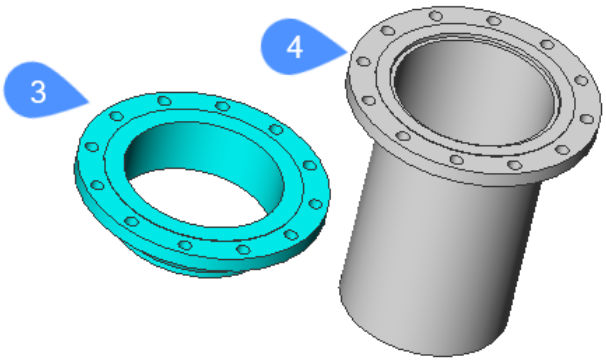
The command automatically creates a set of 3D constraints to correctly connect the parts and maintain their relative position for any further modification.

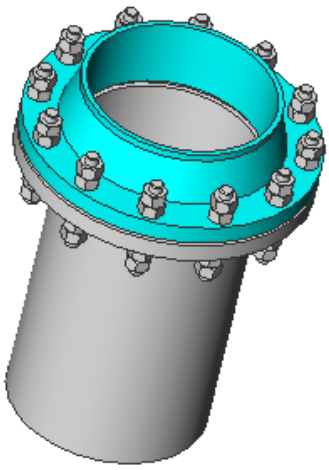


7.115.3 Options within the command

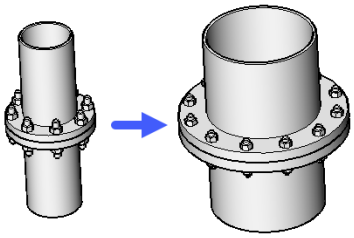
Flip
The next pair of connection point and line is selected for the component (1)

complete flange Assembly
A gasket and a bolting assembly can be selected to automatically connect the flanges (3) and (4).





If the flanges', the gaskets' the bolting assembly's Size and Rating parameters are linked to the global Size and Rating parameters, the flange assembly will be updated properly when these global parameters are changed.



Gasket
Allows to choose a gasket for the flange assembly.

list available gaskets (?)
Lists all available gaskets in the command line.

Path
Allows to choose a custom gasket from the Select a gasket file **Generic Open File** dialog box.

Clear
Removes the chosen gasket.

BOlting assembly
Selects a bolting assembly.

Note: Bolting is a parametric assembly of fasteners for one pair of flanges' bolting holes, e.g., a stud and 4 nuts. All nuts are of the same type. Two of these nuts can be suppressed, according to NutsNumber parameter.

Note: Bolting Assembly is a parametric assembly that contains a polar array of boltings (and connection 2d entities). Its Size and Rating parameters can be linked to the corresponding global parameters.

Note: Flange Assembly is an assembly of the connected pair of flanges, gasket, and bolting assembly.

Path
Allows to choose an existing bolting assembly from the Select a bolting assembly file **Generic Open File** dialog box.

Generate
Generates a boltings assembly from a bolting.

Path
Allows to choose an existing bolting from the **Select a bolting file Generic Open File** dialog box.

Generate
Generates a bolting.

Select a stud

list available studs (?)
Lists all available studs in the command line.

Path
Allows to choose a custom stud from the **Select a stud file Generic Open File** dialog box.

Select a nut

list available nuts (?)
Lists all available nuts in the command line

Path
Allows to choose a custom nut from the **Select a nut file Generic Open File** dialog box.

Specify the bolting name
Gives a name to the bolting generated above and save it in the Select the bolting file dialog box.

Specify the bolting assembly name
Gives a name to the bolting assembly generated above and save it in the **Select the bolting file** dialog box.

Keep the bolting file
Specifies if keep the bolting file

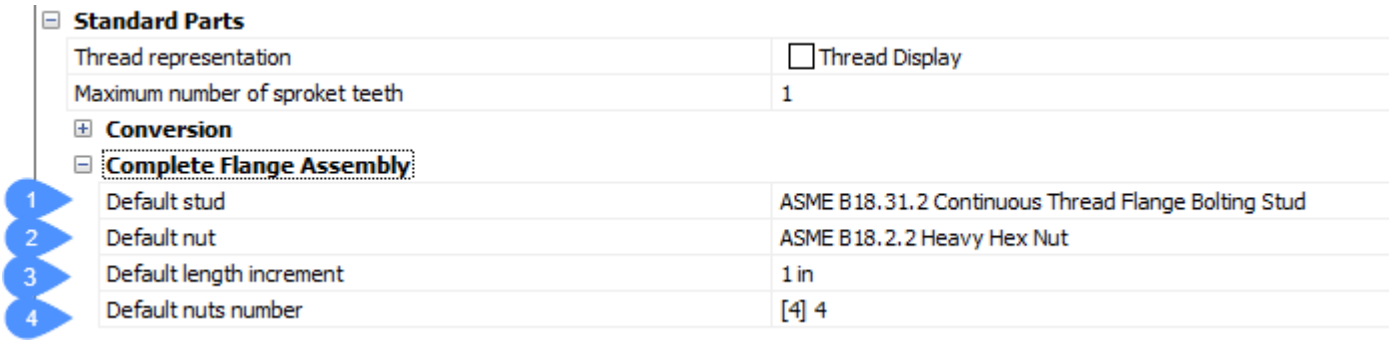
Select nuts number
Specifies the nuts number.

Input length increment
Specifies the length increment of the stud.

Auto
Applies the default stud, nut, length increment, nuts number for the boltings assembly.
A temporary bolting assembly is generated and is chosen. Its file will be removed at the end of the command execution.

Clear
Removes the chosen bolting assembly.

SETtings
Opens the **Settings** dialog box:



1 BoltingAsmDefaultStud - default stud to generate bolting assembly.

2 BoltingAsmDefaultNut - default nut to generate bolting assembly.

3 BoltingAsmDefaultLengthIncrement - default length increment for default stud.

4 BoltingAsmDefaultNutsNumber - default nuts number for bolting assembly.

Back
Returns to the previous command line options.

Finish
Creates the assembly of flanges, gasket and bolting assembly.

7.116 BMCONVERT command

Converts BIM Linear solids with circular profiles to Mechanical assemblies.

Shape Lite Pro Mechanical BIM

7.116.1 Method

Select entities to convert and choose the proper styles for pipes.

7.116.2 Options within the command

Pipe
Allows you to select a style for the pipe segment.

Elbow
Allows you to change the styles for the pipe elbows (45 deg and 90 deg).

Reducer
Allows you to change the styles for the pipe reducers (concentric and eccentric).

Splitter
Allows you to change the styles for the pipe tee and pipe cross.

?
Lists available styles

Note: Use **;** to select multiple styles.

Use current
Uses the current style.

SETtings
Opens the **Settings** dialog box with the **Conversion** section expanded.

7.117 BMCREATECOMPONENT command

Creates library blocks.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The BMCREATECOMPONENT command has been replaced by the CREATELIBRARYBLOCK command.

7.118 -BMCREATECOMPONENT command

Creates and categorizes components via Command line.

Shape Lite Pro Mechanical BIM

Note: The -BMCREATECOMPONENT command has been replaced by the -CREATELIBRARYBLOCK command.

7.119 BMDEPENDENCIES command

Lists all files, containing component definitions inserted in the assembly, in the Command line.

Shape Lite Pro Mechanical BIM

Icon: 

7.119.1 Method

The files are automatically listed in the Command line once the command is run.

7.120 BMDISSOLVE command

Dissolves a mechanical component inserted in the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.120.1 Description

Dissolves a mechanical component inserted in the current drawing by selecting all entities to dissolve manually.

Note: Only components at the first level, directly branching from the root component, in the mechanical hierarchy can be dissolved. To dissolve a component at a lower level, its parent component(s) must be dissolved first.

All subcomponents of a dissolved component are moved up one level in the mechanical hierarchy.

Dissolving a component insert does not affect the source drawing of the component.

The selection set must contain component inserts only. If not, a message displays in the Command line: "No components selected."

7.121 BMEXPLODE command

Creates a block with a representation of the current assembly.

Shape Lite Pro Mechanical BIM

Icon: 

7.121.1 Description

Creates a block with a representation of the current assembly. This block can be inserted at any location.

Note:

- Local mechanical components based on blocks and solids, and mechanical external references are fully supported in exploded views.

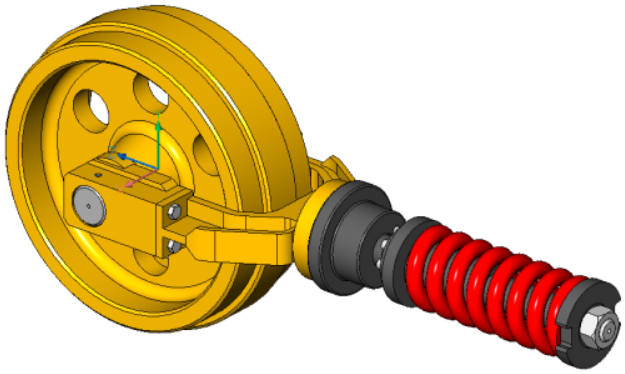
- Non-mechanical entities are also supported, making it possible to create exploded views without creating a mechanical assembly first.

7.121.2 Method

You can create an exploded view using the entire model as selection set or specify a set of parts to create an exploded view from.

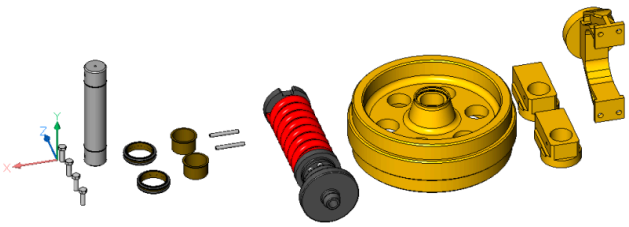
Two levels of the representation are supported: top and bottom. Top-level means that the assembly will be exploded down to top level components. Bottom-level means that the assembly will be exploded down to bottom level components.

The steps to create a representation are automatically completed and are displayed in the **Mechanical Browser**. You can add, delete and reorder the steps from the **Mechanical Browser**. Animate a single step of an explosion or the entire sequence.

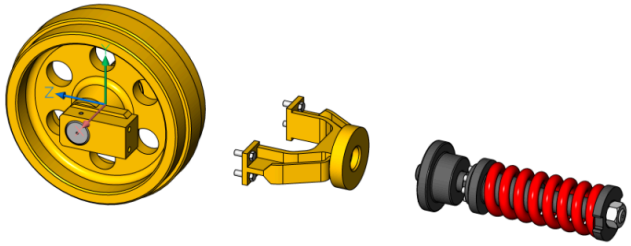


7.121.3 Options within the command

Table by types
Groups identical parts on the same row and different parts on different rows.



Linear
Automatically creates representations of assemblies in a given direction, taking into account possible physical collisions between components.



2Points
Allows you to define or select the linear representation's direction.

Object
Allows you to select an axial entity and sets the direction parallel to that entity.

Last
Uses the previous axis as the representation direction.

View
Allows you to specify a point on view direction axis.

Xaxis
Sets the representation direction parallel to the X axis of the current UCS.

Yaxis
Sets the representation direction parallel to the Y axis of the current UCS.

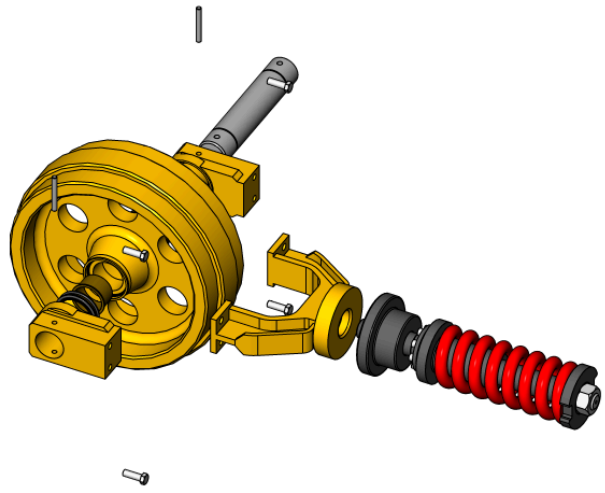
Zaxis
Sets the representation direction parallel to the Z axis of the current UCS.

Automatic
Automatically determines the directions for each part, taking into account possible physical collisions between components.

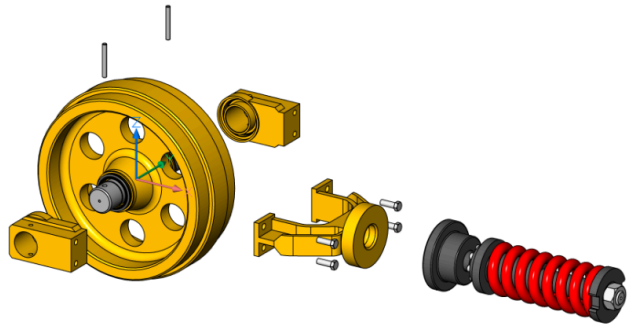
enable Leader follower
Enables the detection of parts that should move in a coherent manner. If enabled, parts that are inserted into other parts will follow the movement of the base parts they are inserted into.

use Current projection for gap
Enables the gap calculation with respect to the current camera orientation. It minimizes the probability that some parts will be hidden by other parts on a 2D view with the current camera orientation.

Auto
Enables automatic gap calculation.



Manual
Selects the manual mode. This creates an exact copy of the assembly, ready to create a customized exploded view.



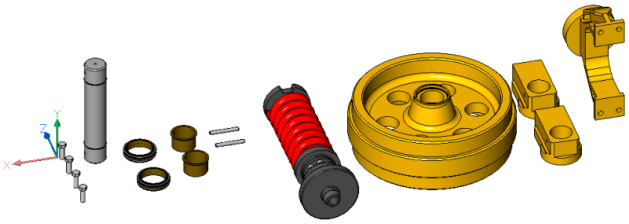
Update
Updates solids and block references in exploded view block with model space objects.

Convert to new format
Converts old format representation to the new one.

Note: This option is available only when there are old format representations in the drawing.

Settings
Gives access to the command settings.

Top
If selected, the assembly will be exploded using only top-level components.



Bottom
If selected, the assembly will be exploded down to bottom-level components.



set Name
Sets the name for the representation.

set Mode
Lets you change the selection filter mode in order to include or exclude non-mechanical entities.

Note: The default selection filter mode is set by the BOMFILTERSETTINGS system variable whose default value is to include only mechanical components, blocks and solids.

ExClude components/Components
Excludes/includes non-mechanical local components, blocks and solids.

Xrefs/Xrefs as Transparent
Doesn't treat/treats external references as transparent.

Xrefs as Transparent/EXclude xrefs
Includes/excludes non-mechanical external references.

Others/Exclude Others
Includes/excludes solids and block references.

Ignore bom status/Respect bom status
Ignores/respects the BOM status.

Transparent Leaf Parts/Stop at Leaf Parts
Includes/excludes non-mechanical objects inside components to appear in the Bottom level representations.

Note: If the **Stop at Leaf Parts** (SLF) flag is ON, bottom level components will be treated as a single object, even if they consist of several solids. If the **Transparent Leaf Parts** (TLF) flag is ON, each solid can be moved independently in the Bottom level representations.

Note: To switch the flag, type accordingly **TLP** or **SLP** in the Command line.

BAck
Returns to the previous prompt.

Edit
Opens the representation for editing.

Note: This is the default option.

Generate drawing views
Generates drafting views from a representation.

Finish
Concludes the command.

7.122 BMEXPLODEMOVE command

Moves selected parts to form an exploded representation in a given direction.

Shape Lite Pro Mechanical BIM

Icon: 

7.122.1 Description

Moves selected parts to form an exploded representation in a given direction, taking into account possible physical collisions between components (similar to the Linear option of the BMEXPLODE command).

This command also has an automatic mode. This determines the direction for each selected part automatically (similar to the Automatic option of the BMEXPLODE command).

7.122.2 Options within the command

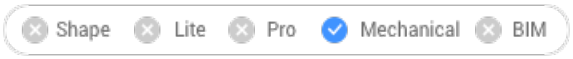
Automatic
Enables automatic mode. An automatically exploded representation of the selected parts is created.

Entire model
Selects the entire model.

Linear
Enables the linear explode mode.

7.123 BMEXPLODESTEPEDIT command

Manages the representation steps.



7.123.1 Description

Allows you to create, delete, merge, copy, split, and reorder representation steps, as well as save and change the current step.

7.123.2 Method

Select a representation created with BMEXPLODE command and open it for edit (from **Mechanical Browser** or with the -BEDIT command).

Perform the necessary operations to get the desired representation following the below options. Do not forget to save the current step.

Then, save changes using the BCLOSE command with the option **Save**.

7.123.3 Options within the command

Add step
Adds a new step after the last one. The new step will be automatically set as current. All unsaved modifications of the representation will be lost.

cOpy step
Copies an existing step.

Note: Step names are case sensitive.

merGe
Merges a step with other specified steps, regardless of their positions.

Move step
Moves a step up (-) or down (+) in the list.

Remove step
Removes a step as well as associated movements of parts. The positions of parts in the representation will be updated accordingly.

reName step
Renames a step.

Save current state
Saves the current positions of parts in the current step.

set CAMera position
Associates an exploded view step with a camera position that will be automatically set during animation and used as default orientations for generated views for this particular step. If no specific camera position is associated with a step, it will inherit the camera position from the initial step. For newly created exploded views, the initial step will get the same camera position as the model.

Set current camera position
Sets the current camera position for the specified step.

Reset camera
Resets the camera position for the specified step.

set Current step
Sets the step as the current and updates the parts in the exploded representation according to this step.

sPlit step
Splits the step to a sequence of steps, each of them corresponding to exactly one part.

Hide parts
Hides the parts that is not involved in the current step.

SHow parts
Shows all parts of the current step.

View options
Sets a view option.

Direction
Sets the direction for animation.

Assembling
Sets direction for animation as assembling.

Disassembling
Sets direction for animation as disassembling.

autoHide
Enables or disables the auto-hide of the parts not involved into a step.

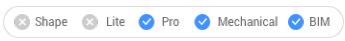
duRation
Sets the default step duration in milliseconds.

Interval
Sets the interval between steps in milliseconds.

list steps (?)
Lists all steps.

7.124 BMEXTERNALIZE command

Converts local components (Mechanical blocks) to external components.



Icon: 

7.124.1 Description

Local components (Mechanical blocks) are converted to external components with the **Save Mechanical Component** dialog box.

The external component drawing will inherit the template visual style, layers, and layouts from the parent document.

Local Mechanical blocks are converted to external references with the **Save Block** dialog box.

This procedure is repeated for each selected local component.

7.124.2 Options within the command

ASk for file names
Toggles the option to ask for file names of externalized components or to generate them automatically.

Yes
File names need to be entered.

No
File names are generated automatically.

> ***Note:*** Some symbols used in component names cannot be used in file names and will be automatically replaced with another symbol, according to BMEXTERNALIZEILLEGALSYMBOLS system variable setting.

Entire model
Selects the entire model. The **Save Mechanical Component** dialog box displays for each local component in the assembly.

7.125 BMFORM command

Creates a new mechanical component or a mechanical block, and inserts it into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.125.1 Description

Creates a new mechanical component as an internal reference in the current drawing. The selected solid(s) are removed from the current drawing.

> ***Note:*** When the MECHANICALBLOCKS system variable is enabled (set to 1), BMFORM creates mechanical blocks by default.

> ***Note:*** If necessary, run the BMMECH command first to initialize the mechanical structure in the current drawing.

7.125.2 Options within the command

Local
Creates a new component and inserts it as a block reference into the current drawing.

Note: When MECHANICALBLOCKS system variable is enabled (set to 1), creates an internal mechanical block. Use BEDIT command to edit block definition.

External
Creates a new drawing which contains the selected solid(s) and can be saved with the **Save Drawing As** dialog box. The selected solid(s) are removed from the current drawing. The new drawing is inserted as an external reference in the current drawing.

mechanical Component
Creates a new mechanical component.

mechanical Block
Creates a new mechanical block.

Template settings
Opens the **Settings** dialog box at the BMFORMTEMPLATEPATH system variable to choose a template file.

7.126 BMHARDWARE command

Opens the **Library** panel.

Shape Lite Pro Mechanical BIM

Icon:

7.126.1 Description

Opens the **Library** panel to display it in the current workspace. The **Library** panel appears at the same size and location as before it was closed or collapsed. Like any other dockable panels, the **Library** panel can be either floating, docked or stacked.

7.127 -BMHARDWARE command

Inserts a standard hardware part as a mechanical component in the current drawing.

Shape Lite Pro Mechanical BIM

7.127.1 Method

This command can only be executed via the options in the Command line.

Note: To initialize the mechanical structure in the current drawing, run the BMMECH command.

Note: This command is obsolete and creates non-parametric standard hardware parts. To insert parametric standard hardware, use the **Library** panel. This contains parametric Standard Mechanical Components.

7.127.2 Options within the command

Parametric
Allows to switch on/off the creation of parametric standard parts.

create Custom
Allows to create custom standard parts from prepared tables. Opens the **Select configuration table file** dialog box that allows to select your created .txt file (or files).

Edit inserted entity
Edits the parameters of the selected insert.

SMART insert
Allows to automatically connect a Piping standard part to an existing Piping standard part by creating a proper set of 3D constraints between them and by copying expressions for the parameters of existing inserted part to the new part.

Rotate component
Specifies the rotation angle by entering a value.

set Base point
Specifies a new base point for the insert.

Name
Renames the insert by entering a name.

insertion Type
Specifies the insertion type.

Local
Inserts the part locally.

External
Inserts the part externally.

Flip
Inserts the selected insert in the opposite face of the solid.

mUltiple
Allows you to insert multiple copies by entering multiple insertion points.

Array
Creates an associative array of components.

Change target 3d solids
Allows you to choose the new set of target solids. Any current component-based features will be removed and new ones will be created for the selected solids.

cleaR
Removes the existing component-based features with their geometry. See the BMUNLINK command.

Select all affected 3d solids
Automatically chooses solids. A solid is selected if any solids from the BC_SUBTRACT layer or if any layer from the BC_UNITE touches or intersects it.

7.128 BMHIDE command

Hides an inserted mechanical component.

Shape Lite Pro Mechanical BIM

Icon:

7.128.1 Description

Hides a mechanical component inserted in the current drawing.

The command affects the visibility of the selected inserts only. Hidden inserts will still be considered by commands such as BMBOM, BMMASSPROP etc.

7.128.2 Options within the command

COmponent name
Type the component name of the component insert(s) you want to hide. All inserts of the specified component will be hidden.

Insert name
Type the insert name of the component insert you want to hide.
If you want to hide a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/).

7.129 BMINSERT command

Inserts a component into the current drawing.

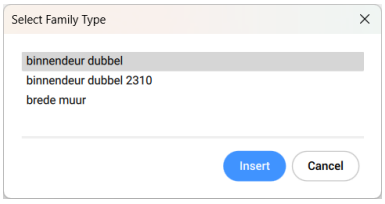
Shape Lite Pro Mechanical BIM

Icon:

7.129.1 Description

Opens the **Select file to insert** dialog box to select a DWG or an RFA file to insert into the current drawing.

When inserting a Revit® family containing multiple types, the **Select Family Type** dialog box displays, which enables you to import the preferred type of the Revit® family. The RFA files attributes are also imported.



After selecting the file and choosing **Open**, the **Properties** panel displays, and you are prompted to the Command line.

Note:

- In order to insert a file with a mechanical block as mechanical block, you must enable MECHANICALBLOCKS system variable.
- A simple file (unmeched) is inserted as a mechanical block when the MECHANICALBLOCKS system variable is enabled.

7.129.2 Method

Specify a point in the current drawing where the entity will be inserted. You can change the insertion before it is inserted using the **Properties** panel and the Command line options.

7.129.3 Options within the command

Edit inserted entity
Enables you to change the parameter expressions for the inserted entity. Continue editing individual parameters until you press Enter to end the option. This option is also available in the **Hotkey Assistant** widget.

SMART insert
Enables you to connect a Piping standard part to an existing Piping standard part. It automatically creates appropriate 3D constraints between the two parts and copies expressions for the parameters of the existing part to the new part. This option is also available in the **Hotkey Assistant** widget.

Rotate component
Enables you to change the rotation angle for the inserted entity.

set Base point
Enables you to change the base point for the inserted entity.

Note: By default is <0,0,0>.

Name
Enables you to change the instance name for the inserted entity.

insertion Type
Enables you to change the insertion type for the inserted entity.

Local
References an entity definition within the current drawing.

External
References an entity definition outside the current drawing.

Note: Mechanical components as well as mechanical blocks can be inserted as external references.

Flip
Enables you to flip the direction for the inserted entity.

mUltiple
Enables you to insert multiple copies of the same entity by specifying an insertion point for each instance or creating an Array.

Note: Continue inserting entities until you press **Enter** to end the command.

Array
Enables you to create an associative array of the inserted entity by specifying the base point, distance between columns, distance between rows and end point of the array.

Direction
Allows you to select an existing axial entity to define the direction.

2Points
Selects two points to define the direction.

Xaxis
Selects the X axis as direction.

Yaxis
Selects the Y axis as direction.

Single row
Distributes entity copies into a single row.

Rectangular
Distributes entity copies into any number of rows.

Accept
Accepts the resulting array.

Change target 3d solids
Enables you to apply the inserted entity to existing 3D solids in the current drawing.

cleaR
Clears the selection set to ensure no solids are affected by the inserted entity.

Select all affected 3d solids
All solids intersecting or touching solids in the BC_SUBTRACT and BC_UNITE layers of the inserted entity are affected.

7.130 -BMINSERT command

Inserts sheet metal form features and components in the current drawing.

Shape Lite Pro Mechanical BIM

7.130.1 Description

Inserts sheet metal form features, mechanical and BIM components in the current drawing.

Note: This command inserts a *.RFA file as a BIM component.

This command operates at the Command line. Enter the path of the file to be inserted and choose one of the options. For more details about the options see BMINSERT command.

7.131 BMLINK command

Changes target 3D solids of component-based features.

Shape Lite Pro Mechanical BIM

Icon:

7.131.1 Description

Changes the target 3D solids of component-based features by adding or removing target 3D solids.

7.131.2 Options within the command

Change target 3d solids
Specifies a set of target solids. Component-based features will be removed and new ones will be created for selected solids.

Append
Adds new solids to set of target solids. Component-based features will be updated and new ones will be created for the selected solids.

cleaR
Removes the existing component-based features and their geometry (see the BmUnlink command).

Select all affected 3d solids
This mode is similar to **Change target 3d solids** but solids are chosen automatically.

Note: A solid will be chosen if any solids from BC_SUBTRACT layer intersects it or if any other solid from the BC_UNITE layer touches or intersects it.

7.132 BMLOCALIZE command

Converts external components to local components.

Shape Lite Pro Mechanical BIM

Icon: 

7.132.1 Description

Converts external components to local components by selecting the components manually.

Note: Mechanical external references are supported as well as traditional mechanical components.

If multiple inserts of the same external component exist in the model, all inserts are converted to local components.

7.132.2 Options within the command

Entire model
Converts all external components in the model to local components.

7.133 BMMASSPROP command

Computes mass properties for the current model using densities of materials assigned to its components.

Shape Lite Pro Mechanical BIM

Icon: 

Note: BmMassProp will format the values of mass, volume and linear nature with their corresponding units only if the PROPUNITS system variable has the corresponding bits set to 1 or the corresponding flag in the Settings dialog for the Property units field is checked.

7.133.1 Method

The density values used for the mass property calculations are taken from materials defined by the Material property of the components and subcomponents, which can be either inherited from the main component of the assembly or set explicitly. The density of the main component is defined by the materials set in the Material field in the Mechanical Browser dialog. If a material does not have an assigned density, or its density is less or equal to zero, an error message displays, and the mass properties will not be computed. All materials involved in the assembly must have non-zero non-negative densities.

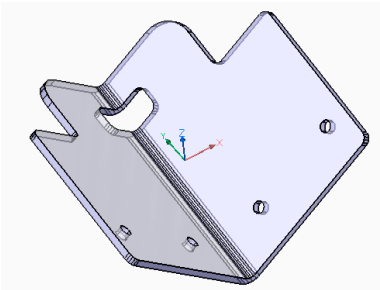
Selects the 3D solids and subcomponents or the entire model and will get a Report in the Prompt History window:

```
---------------- Component ----------------
Mass: 6.7757 kg
Volume: 846.9586 cm³
Centroid: X= -267.9141 mm Y= 46.2431 mm Z= -435.3534 mm
Moments of inertia: X= 1.3564 kg·m² Y= 1.8427 kg·m² Z= 0.5730 kg·m²
Products of inertia:
XY: -0.0808 kg·m²
YZ: -0.1535 kg·m²
ZX: 0.7934 kg·m²
Radii of gyration: X= 447.4159 mm Y= 521.4979 mm Z= 290.802 mm
Principal moments and X-Y-Z directions about centroid:
I: 0.0551 kg·m² along X= 0.0000 Y= -0.7071 Z= 0.7071
J: 0.0570 kg·m² along X= 0.9906 Y= 0.0970 Z= 0.0969
K: 0.0899 kg·m² along X= -0.1371 Y= 0.7004 Z= 0.7004
```

7.133.2 Options within the command

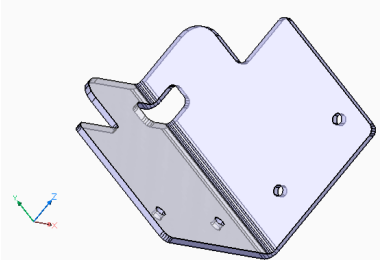
Yes

Aligns the UCS with the principal moment axes.



No

Keeps the current UCS.



Note: The command can be interrupted by pressing the CANCEL button when it takes too long.

7.134 BMMECH command

Converts the current drawing into a mechanical component.

Shape Lite Pro Mechanical BIM

Icon: 

7.134.1 Description

If the drawing contains block references or external references, they can also be converted into inserts of local and external components. If mechanical blocks are enabled, solids and regular named blocks can be converted to mechanical entities and blocks correspondingly as well.

If the drawing is a mechanical component already, BMMECH does nothing.

Note: Mechanical blocks are enabled if the MECHANICALBLOCKS system variable is set to 1.

7.134.2 Options within the command

create Legacy components

(Only if Mechanical blocks are enabled) Changes the command mode to create mechanical (legacy) components.

create mechanical Blocks

(Only if Mechanical blocks are enabled) Changes the command mode to create mechanical blocks and mechanical entities. If there is no root component in the document, the document will become a mechanical block itself.

Yes - to convert all

If the drawing contains block references or external references, and the if the command is in Mechanical blocks mode (MECHANICALBLOCKS system variable is ON), block references, external references and solids in the model space will be converted to mechanical blocks and entities. Also, arrays of block references or external references will be converted into mechanical blocks.

If MECHANICALBLOCKS system variable is OFF, blocks are converted to internal components and external references are converted to external components.

Note: Blocks converted to mechanical (legacy) components are not available in the INSERT command. Use BMINSERT to insert blocks as internal components. However, mechanical blocks are available in the INSERT command, as they are regular blocks with mechanical data attached.

Note: When a BIM entity is converted to a mechanical block, its BIM name is retained as a component name.

Blocks - to convert blocks

(Only if Mechanical blocks are enabled). If the drawing contains block references or external references, they will be converted to mechanical blocks. Solids from the model space will not be converted in that mode.

No - to skip conversion

The mechanical structure is initialized but blocks and external references are not converted.

7.135 BMNEW command

Creates a mechanical component as a new drawing file.

Shape Lite Pro Mechanical BIM

Icon: 

7.135.1 Description

Creates a mechanical component by automatically opening a new drawing file.

Note: When the MECHANICALBLOCKS system variable is enabled (ON), a mechanical root block is created instead.

7.136 BMOPEN command

Opens the source drawing of an external mechanical component.

Shape Lite Pro Mechanical BIM

Icon: 

7.136.1 Description

Opens the source drawing of an external mechanical component by selecting it in the model, in order to change or visualize it.

7.137 BMOPENCOPY command

Opens a copy of a component insert as a new drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.137.1 Description

Opens a copy of a selected external mechanical component in a new drawing.

If you open a copy of a parametric mechanical component, the current values of the various parameters are applied to the copy.

After saving it you may replace the original component with the newly created drawing using BMREPLACE command.

7.138 -BMPARAMETERS command

Allows to list and edit the parameters of inserted components and associative arrays.

Shape Lite Pro Mechanical BIM

7.138.1 Description

Allows to list and edit the parameters of inserted components and associative arrays. By pressing Enter, the parameter expression can be changed.

Note: Parameters of components nested inside an array can be changed or assigned to an expression.

7.138.2 Options within the command

Edit
Edits the value of a parameter.

Delete
Allows to reset a previously edited parameter to its default value.

Link to parameter
Assigns parameters as an expression for a parameter of a dimensional constraint (both 2D and 3D) associated with an entity which has been put into an associative array.

?
Lists the parameter, expression and value of the selected components in the Prompt History window.

Note: Press F2 to display the Prompt History window.

7.139 BMPROPERTIES command

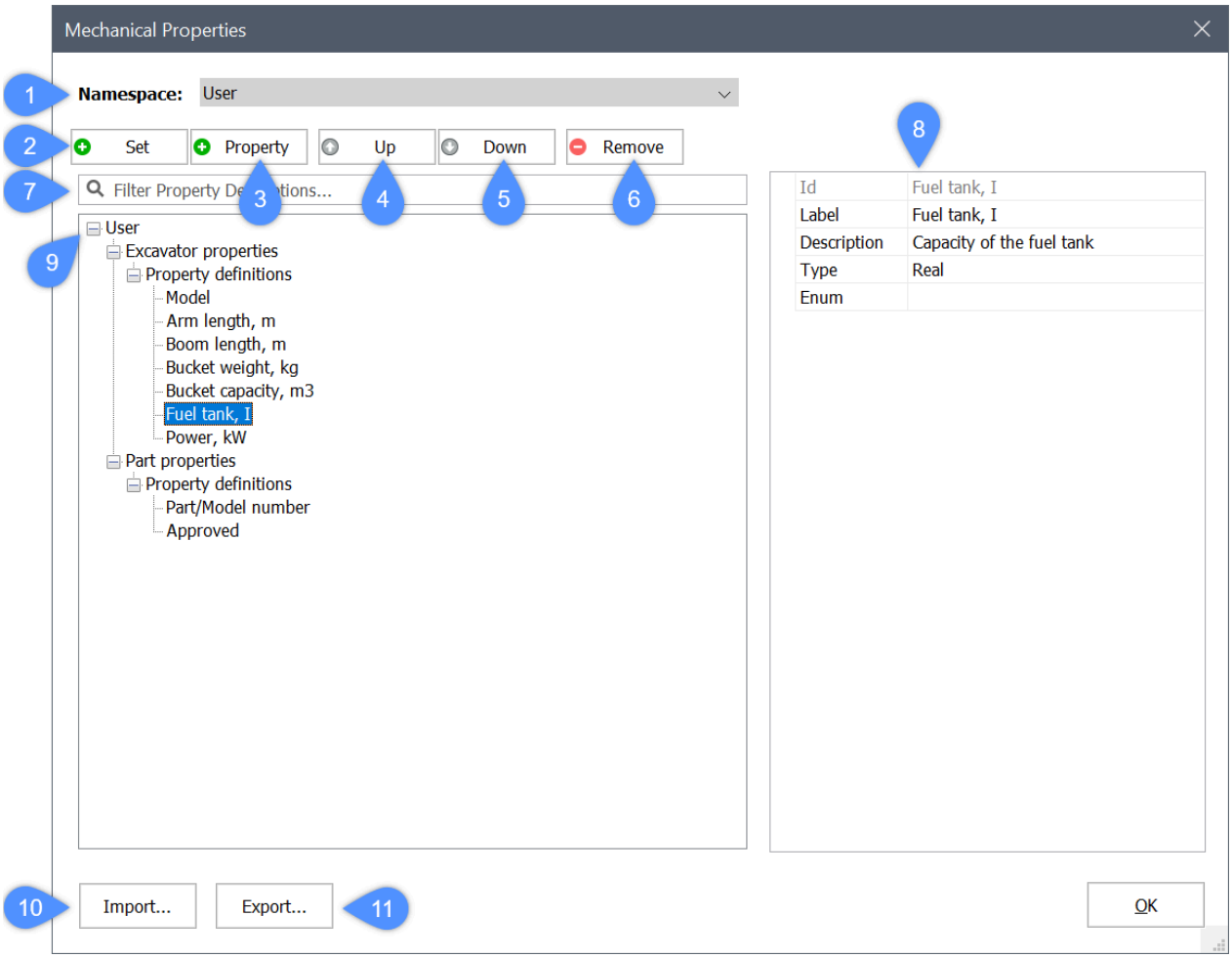
Creates, edits and deletes property definitions and organizes properties in property sets.

Shape Lite Pro Mechanical BIM

Icon: 

7.139.1 Description

Opens the **Mechanical Properties** dialog box to create and manage property definitions.



1 Namespace

2 Add Set

3 Add Property

4 Move up button

5 Move down button

6 Remove

7 Filter

8 Properties

9 Properties Tree

10 Import

11 Export

7.139.2 Namespace

For mechanical component, currently one namespace is available:

User
Allows you to create user defined properties.

7.139.3 Add Set

Allows you to add a property set in the current namespace.

7.139.4 Add Property

Allows you to add a property to the currently selected property set.

7.139.5 Remove

Deletes the selected item.

7.139.6 Move up button

Moves up the selected property or value.

7.139.7 Move down button

Moves down the selected property or value.

7.139.8 Filter

Displays only the properties of which the **Id** and **Label** contain the search string typed. The properties tree is filtered as you type.

7.139.9 Properties

Displays the properties of the selected property sets, property definitions and value definitions.

7.139.10 Properties Tree

Displays the property sets, property definitions and value definitions.

7.139.11 Import

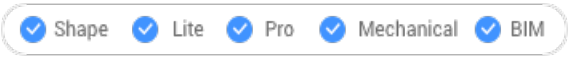
Opens the **Select XML file to import** dialog box.

7.139.12 Export

Opens the **Select location to save xml file** dialog box.

7.140 BMPOUT command

Saves the current view as a BMP file.



7.140.1 Description

Opens the **Save Bitmap** dialog box to save the current view (model or paper space) as a bitmap BMP file.

7.141 BMRECOVER command

Recovers a broken mechanical structure.

Shape Lite Pro Mechanical BIM

7.141.1 Description

Recovers a broken mechanical structure selected through the **Select mechanical component** dialog box. The command overwrites the original drawing.

Note: The command differs from the basic AUDIT and RECOVER commands. It is recommended to use BMRECOVER to mend issues with a MCAD drawing.

7.142 BMREPLACE command

Replaces a component insert.

Shape Lite Pro Mechanical BIM

7.142.1 Description

Replaces a mechanical component inserted in the current drawing with another one selected from **Select file for component** dialog box.

Note:

- Mechanical blocks can also be replaced by other mechanical blocks.
- The command does not support cross-replacement: each kind of component can only be replaced by a component of the same kind.

7.142.2 Options within the command

SIMilar inserts

Allows you to replace similar inserts automatically, depending on the chosen option.

- **Yes**: replaces all similar inserts of the selected mechanical component(s) in assembly.
- **No**: replaces the selected component(s) only.

component Type

Allows you to choose the insertion type for a replacement component. The following options are available:

- **Local**: the new component will be local.
- **External**: the new component will be external.
- **Keep as is**: the new component inherits the insertion type from the replaced component.
- **component Default**: the new component will be linked as specified in its source file.

See External and local components in the User Guide.

Parameter changes

Allows you to control which parameter values will be used after the replacement is complete.

- **Yes - reapply**: parameter values of the replaced component are applied.
- **No - use replacement as is**: parameter values of the replacement component are applied.

from FIle

Allows you to select a DWG file from the **Select file for component** dialog box to choose the replacement component.

7.143 BSCALE command (Express Tools)

Scales block references relative to their insertion points.

Shape Lite Pro Mechanical BIM

7.143.1 Options within the command

Absolute
Specifies an absolute scale factor in X and Y-direction or in X-, Y- and Z-direction.

Relative
Specifies a relative scale factor in the X and Y-direction or in the X-, Y- and Z-direction.

7.144 BMEXPLODECONFIG command

Assigns preferable directions for entities during automatic generation of exploded views.

Shape Lite Pro Mechanical BIM

7.144.1 Description

Automatically converts directions defined in WCS to local directions of selected entities during generation of exploded views.

7.144.2 Method

Select entities which will use the preferable direction for exploded view, then set the direction.

7.144.3 Options within the command

Set
Selects entities to set preferable direction for disassembly.

Root part
Selects only root part.

Entire model
Selects entire model.

Remove
Selects entities to remove from preferable direction for disassembly.

Select the direction
Selects the direction for selected entities which will be follow on exploded views.

Select axial entity
Allows you to select an existing axial entity.

2Points
Selects two points to define the direction.

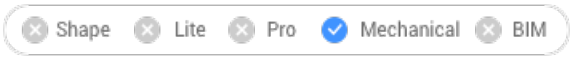
Xaxis
Selects the X axis as direction.

Yaxis
Selects the Y axis as direction.

Zaxis
Selects the Y axis as direction.

7.145 BMSEQUENCE command (Experimental)

Creates and edits sequences for your assembly design.



7.145.1 Disclaimer



Sequence Manager is an experimental feature that may not yet be stable and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

7.145.2 Method

There are two methods for operating the command, depending on whether the **Experimental mode** is enabled or disabled.

1 When **Experimental mode** is disabled:

Select one or more entities for a step and press **Enter**, and so on, in the order of disassembly. Do this till the end step. After the entire assembly sequence is specified, enter a unique name for the assembly sequence. You can finish the sequence at any time.

Note: Press **CTRL+Z** to local undo to previous assembly step.

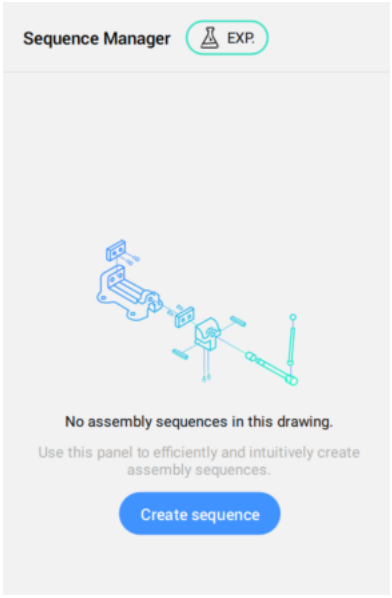
2 When **Experimental mode** is enabled:

Select the entities, enter a unique name for the assembly sequence, and the **Sequence Manager** panel opens to edit the steps of the sequence.

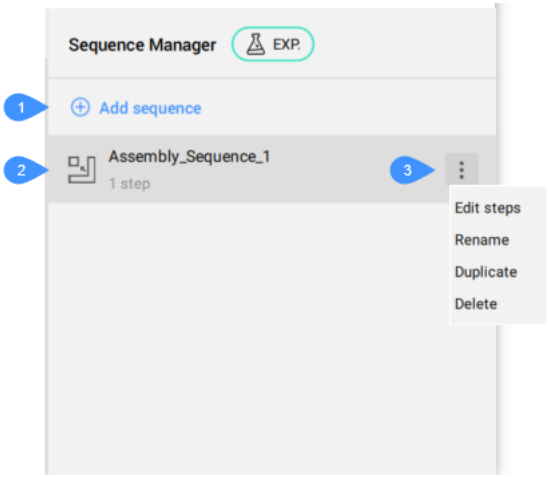
Note: Only mechanical instances can be selected.

The **Sequence Manager** panel allows you to manage the sequences and exploded views of a mechanical assembly.

If there are no assembly sequences in the drawing, there is only **Create sequence** option in the panel.



You can create the first sequence in the drawing by pressing the **Create sequence button**. The default sequence name is **Assembly_Sequence_1**.



1 Add sequence

2 Assembly_Sequence created

3 Options button

Add sequence
Adds a Assembly_Sequence in the list.

Assembly_Sequence created
Displays the created Assembly_Sequence.

Options button
Displays a list with editing options for the Assembly_Sequence.

Edit steps
Opens the **Sequence Manager** panel in the block-edit mode of the block in which the Assembly_Sequence is stored.

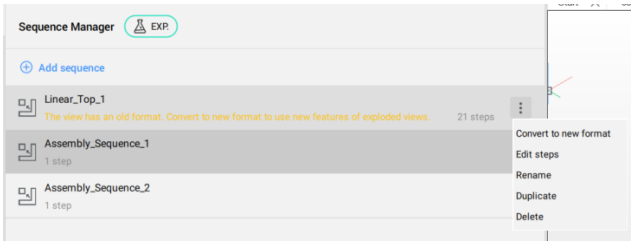
Note: Also, double click an Assembly_Sequence opens the **Sequence Manager** panel in the block-edit mode.

Rename
Displays the **Rename sequence** dialog box to introduce the new name for the Assembly_Sequence.

Duplicate
Creates a duplicate for the Assembly_Sequence.

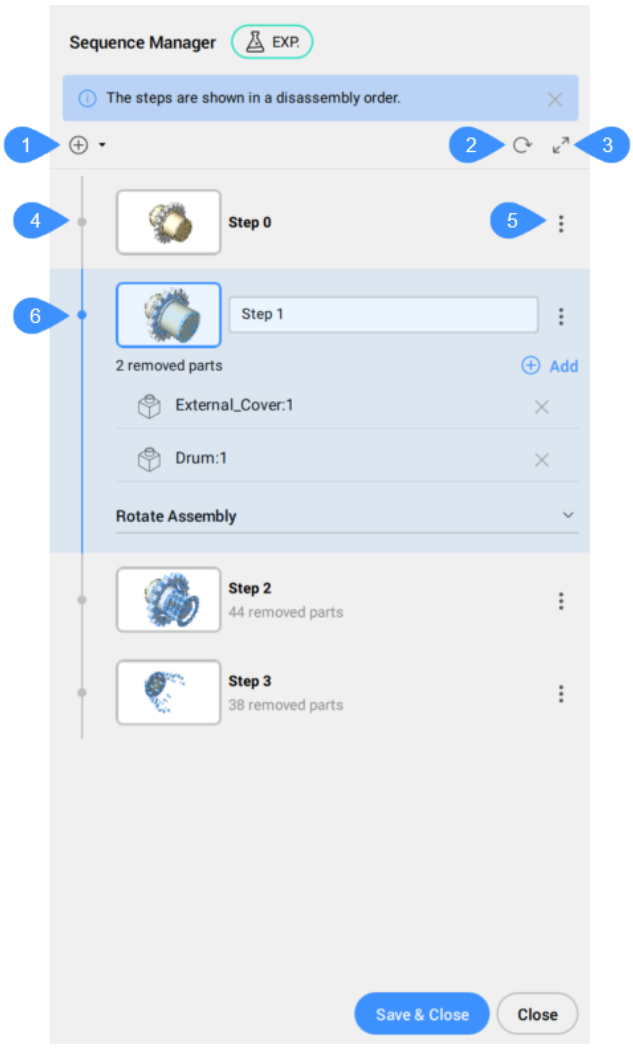
Delete
Deletes the Assembly_Sequence.

Exploded views in the old format can be converted to the new format by choosing the **Convert to new format** option from the **Options button**.



When open the **Sequence Manager** panel in the block-edit mode of the block in which the sequence is stored, you are able to manage the sequence steps.

The **Sequence Manager** panel in this mode shows a list of steps in the sequence in disassembly order, with a thumbnail for each step and a list of all parts in that step.



1 Add new step

2 Update thumbnails

3 Switch to large thumbnails

4 List of steps

5 Options button

6 Current/active step

Add new step
Adds a new step after the last step.
The drop down list contains several options for adding the step:

After last step
Adds a step after the last step.

After current step
Adds a step after the current step.

Before current step
Adds a step before the current step.

Before first step
Adds a step before the first step.

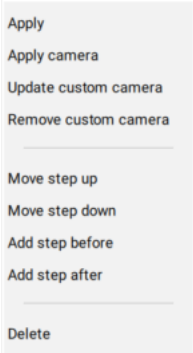
Update thumbnails
Updates the thumbnails after editing step(s).

Switch to large thumbnails
Enables to display large thumbnails.

List of steps
Displays the steps of the sequence.

Note: Set the step as the current by clicking on its thumbnail.

Options
Lists the options applicable to the step.



Apply
Sets the step as current.

Apply camera
Applies the camera position for the step.

Update custom camera
Updates the custom camera position.

Remove custom camera
Removes the custom camera position.

Move step up
Moves the step up.

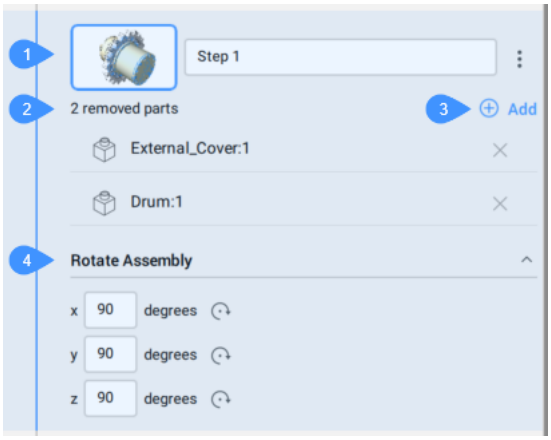
Move step down
Moves the step down.

Add step before
Adds a step before this one.

Add step after
Adds a step after this one.

Delete
Deletes the step.

Current/active step
Allows you to edit the current step.



1 Thumbnail

2 Removed parts list

3 Add removed part

4 Rotate Assembly

Thumbnail
Displays the thumbnail of the step.

Note: By clicking on the thumbnail you will launch the **Apply camera** option, to apply the custom camera.

Removed parts list
Lists the entities (mechanical components) that were disassembled in this step.

Note: There are **Show more** and **Show less** buttons for large lists to manage the display of components.

Add removed part
Adds one or more mechanical components to this step.

Rotate Assembly
Rotates the assembly around the X, Y, or Z axes. You can click on the thumbnail to keep the camera position (**Apply camera**).

The assembly sequence is registered as an exploded view block under **Representations** in the **Mechanical Browser** panel. Step number is descending so that the numbering is ascending in the BMASSEMBLYINSPECT command. The sequence to the exploded view block is reversed to represent assembly sequence or disassembly sequence.

Note: The sequence can be inspected with BMASSEMBLYINSPECT command.

7.145.3 Options within the command

When **Experimental mode** is disabled:

Finish
Finishes the creation of the sequence.

When **Experimental mode** is enabled:

Entire drawing
Selects entire drawing.

7.146 BMSEQUENCEPANELCLOSE command

Closes the **Sequence Manager** panel.

Shape Lite Pro Mechanical BIM

7.146.1 Description

Closes the **Sequence Manager** panel to hide it from the current workspace. If the **Sequence Manager** panel is stacked when you close it, the **Sequence Manager** tab or icon is removed from the stack.

7.147 BMSEQUENCEPANELOPEN command

Opens the **Sequence Manager** panel.

Shape Lite Pro Mechanical BIM

7.147.1 Description

Opens the **Sequence Manager** panel to display it in the current workspace. The **Sequence Manager** panel appears at the same size and location as it did before it was closed or collapsed. Like any other dockable panel, the **Sequence Manager** panel can be either floating, docked or stacked.

7.147.2 Disclaimer

EXP.

Sequence Manager is an experimental feature that may not yet be stable and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

7.148 BMSEQUENCESTEPEDIT command

Edits a sequence step by adding or removing parts from it.

Shape Lite Pro Mechanical BIM

7.148.1 Method

Note: The command is available only for editing a step from a sequence.

7.148.2 Disclaimer

EXP.

Sequence Manager is an experimental feature that may not yet be stable and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

7.148.3 Options within the command

Add part
Add parts to the selected step of the sequence.

Remove part
Removes parts to the selected step of the sequence.

7.149 BMSHOW command

Shows hidden mechanical components from the drawing.

Shape Lite Pro Mechanical BIM

7.149.1 Description

Shows previously hidden mechanical components inserted in the current drawing.

7.149.2 Options within the command

show ALl
Shows all component inserts.

COmponent name
Type the component name of the component insert(s) you want to show. This affects all inserts of the specified component.

Insert name
Type the insert name of the component insert you want to show.
If you want to show a nested component insert (subcomponent insert), first type the insert name of the parent component insert, then the component insert name separated by a forward slash (/).

7.150 BMTRAILINGLINES command

Creates all necessary trailing lines for selected parts.

Shape Lite Pro Mechanical BIM

7.150.1 Description

Creates all necessary trailing lines by selecting entities.

When you select two parts and a 3D polyline, a custom trailing line based on the 3D polyline will be generated. This trailing line will be visible in the **Mechanical Browser**.

Note: An exploded view needs to be selected and open for editing before the command can be run.

7.150.2 Options within the command

use Origin points
Switches the mode used to choose the reference point for parts to the origin (insertion point) of the component.

use Center points
Switches the mode used to choose the reference point for parts to the center point of the bounding box of the component.

Entire model
Selects the entire model and starts the processing.

for current Step
Generates a trailing line for the current step, which means it will be shown only for this step.

Note: By default, trailing lines are generated for the current step.

7.151 BMUNLINK command

Breaks the connection between a component insert and a 3D solid retaining the opening.

Shape Lite Pro Mechanical BIM

7.151.1 Description

Breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening by selecting the components.

7.152 BMUNMECH command

Converts mechanical components.

Shape Lite Pro Mechanical BIM

7.152.1 Description

The command converts the current mechanical component into a plain drawing.

Note: The command applies to drawings that are a mechanical component only.

7.152.2 Options within the command

Yes
Removes the mechanical structure from the drawing, by turning it into a plain drawing.
Inserts of local components are converted to block references (the component name becomes the block name). Inserts of external components are converted to external references.

No
Aborts the command.

7.153 BMUPDATE command

Reloads all referenced components from external files and updates BOM tables.

Shape Lite Pro Mechanical BIM

Icon: 

7.153.1 Description

The command:

- Reloads referenced components from external files (BmInsert).
- Rebuilds component based features, sheet metal form features and BIM windows/doors components (see BC_SUBTRACT layer, Form Features, BmInsert).
- Updates the Bill Of Material (BmBom).
- Regenerates drawing views produced out of 3D model (ViewBase).

Note: If the BMAUTOUPDATE system variable is ON, external assembly components are reloaded on opening the file.

7.153.2 Options within the command

MOde
Defines the new placement face.

Note: Each inserted component-based feature or BIM Window component has an associated placement face of the correspondent 3D solid the feature is attached to. This is the face which will be used for feature placement in the BMINSERT command. The associated placement face of the feature can be lost in modeling or copying operations or it can be absent if the feature has been inserted in an empty area. The BMUPDATE command allows to re-associate the feature with its placement face.

Automatic
The placement face for the feature is detected automatically.

Manual
Select a new placement face manually.

Entire model
Updates the entire model.

7.154 BMVSTYLE command

Applies a visual style to a mechanical component inserted.

Shape Lite Pro Mechanical BIM

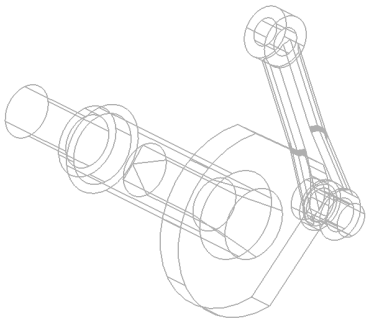
7.154.1 Description

Applies a chosen visual style to all or only some inserted components by typing ALL or the component's names inserted, separated by slashes (/). Component insert names are case sensitive.

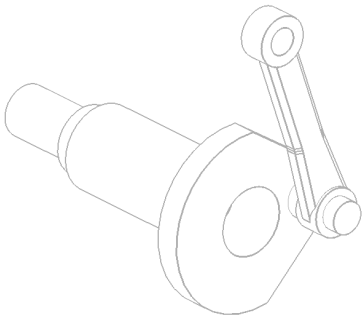
7.154.2 Options within the command

by Viewport
Applies the current visual style of the viewport.

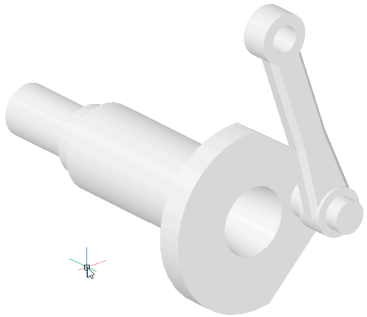
Wireframe
Applies the 3D Wireframe visual style.



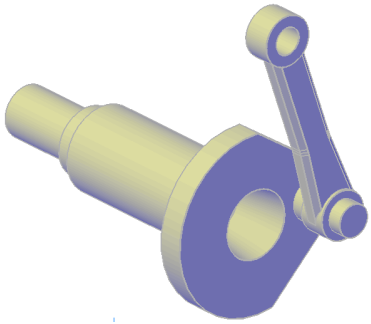
Hidden
Applies the Hidden visual style.



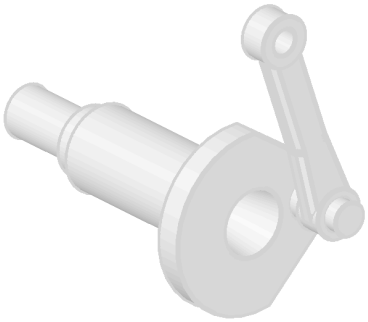
Realistic
Applies the Realistic visual style.



Conceptual
Applies the Conceptual visual style.



Modeling
Applies the Modeling visual style.



by Name
Applies a named visual style.

7.155 BMWELDANNOTATE command

Creates annotates for arbitrary 2D entity in the paper space with caterpillars and welding symbols.

Shape Lite Pro Mechanical BIM

Icon:

7.155.1 Method

Select the welding symbol type and the curve from the paper space to annotate.

7.155.2 Option within the command

Front
Creates a caterpillar welding symbol. Select the symbol representation from the **Load Linetypes** dialog box.

Note: If the lines are already loaded, a warning message box is opened to agree with the replacement of lines.

Load linetype
Opens the **Load Linetypes** dialog box to select another symbol representation.

Select curve
Applies the caterpillar symbol on the selected curve.

Note: A preview of the caterpillar symbol is shown.

Partial
Defines a part of the curve to apply the caterpillar symbol.

Flip
Flips the caterpillar symbol side for the last selected curve.

Scale
Scales the caterpillar symbol for the last selected curve.

Side
Creates a side view welding symbol.

Fillet
Creates a fillet side view symbol.

J-groove
Creates a J-groove side view symbol.

V-groove
Creates a V-groove side view symbol.

U-groove
Creates a U-groove side view symbol.

Seam
Creates a seam side view symbol.

7.156 BMWELDING command

Creates a fillet or groove welding for faces (or two sets of faces) of different 3D solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.156.1 Method

Select the faces of the two 3D solids (members) between which the weld will be created, if possible:

- For fillet welding you can select a face of the first member and one or multiple faces of the second member.
- For groove welding you can select one or multiple faces of the first member and one or multiple faces of the second member.

Note: When the operation is not possible for all selected faces, a notifying balloon will appear on the screen. Click **More details...** to open the **Report** panel where the faces not included in the operation are listed.

7.156.2 Options within the command

Fillet
Creates a fillet welding using one of the following sizes, according with the value of the FILLETWELDING-COMBINEADJACENT system variable:

Input Z
Specifies a value for the Z (Leg) length of the fillet welding.

Note: The default Z size is stored by the FILLETWELDINGZSIZE system variable.

Input A
Specifies a value for the A (Throat) size of the fillet welding.

Note: The default A size is calculated as FILLETWELDINGSIZE value * sin(45 degrees).

Groove
Creates a groove welding.

Note: Welding features are added in the **Mechanical Browser** panel. The context menu for a welding feature contains **Add weld symbol properties** option that opens **Weld Symbol** dialog box (see the related article **Weld Symbol dialog box**). The weld symbol properties can be edited or deleted. After this, in the drawing views (Paper Space), the corresponding weld symbols can be created using the BMWELDSYMRETRIEVE command.

7.157 BMWELDSYMRETRIEVE command

Places welding symbols added to a welding feature in the drawing views.

7.157.1 Description

Retrieves weld information (welding symbols) from 3D weld bodies that contain welding features. Weld information is created with **Add weld symbol properties** option from the context menu of a welding feature from the **Mechanical Browser** panel.

Note: This command is available only in Paper Space.

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

7.157.2 Method

Select weld solid to retrieve its welding symbol.

7.157.3 Options within the command

Auto
Retrieve welding symbols created for all visible weld solids of the selected viewports.

7.158 BMXCONVERT command

Converts X-Hardware solids to mechanical components.

Shape Lite Pro Mechanical BIM

7.158.1 Description

Converts X-Hardware solids in the current drawing to mechanical components.

7.159 BOUNDARY command

Creates closed polylines from bounding entities.

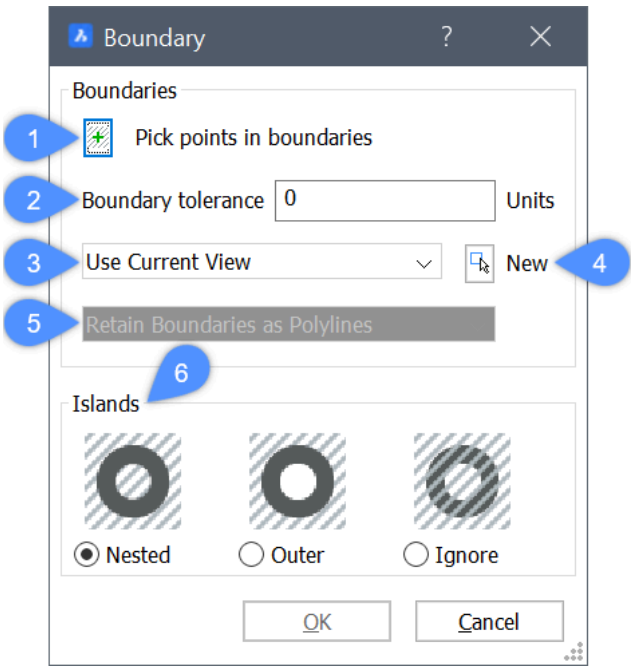
Shape Lite Pro Mechanical BIM

Icon: 

Alias: BO, BPOLY

7.159.1 Method

Begin creating boundaries by selecting options in the **Boundary** dialog box. Options allow you to pick an internal point, specify the boundary set, and detect islands.



1 Pick points in boundaries

2 Boundary tolerance

3 Boundary set

4 New

5 Retain Boundaries as Polylines

6 Islands

Pick points in boundaries
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.

Boundary tolerance
Specifies the largest gap that can exist in the boundary for BricsCAD to consider the area closed.

Note: When boundary tolerance is 0, no gaps can exist.

Boundary set
Specifies 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.

New
Allows you to create a new selection set of entities that make up the boundary. It temporarily dismisses the **Boundary** dialog box.

Retain Boundaries as Polylines
Indicates how boundaries are retained. This option is read only.

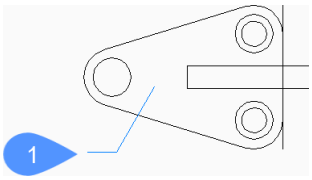
Islands
Specifies island detection. An island is an enclosed area inside a boundary.

Nested
Treats every island as a boundary.

Outer
Creates a boundary only of the outermost entities.

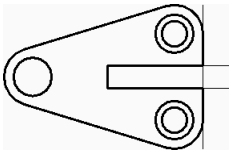
Ignore
Ignores the innermost areas. A boundary is created between the outermost area and islands.

Original entities:



1 Internal point

Resulting polylines:



7.159.2 Options within the command

Select entities
Allows you to select entities to use as boundaries.

Undo
Undoes the last internal pick point and continue picking points to specify additional enclosed areas.

7.160 -BOUNDARY command

Creates closed polylines from bounding entities.

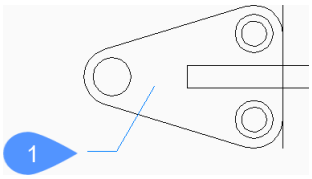
Shape Lite Pro Mechanical BIM

Alias: -BO

7.160.1 Description

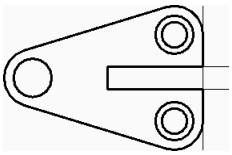
Create closed polylines, defined by surrounding entities using the Command line. Options allow you to pick an internal point, specify the boundary set, and detect islands. See the BOUNDARY command to access similar functionality using a dialog box.

Original entities:



1 Internal point

Resulting polylines:



7.160.2 Methods to create a boundary

There is one method to begin creating a boundary:

- Internal point

Internal point
Begin creating boundaries by specifying a point inside an enclosed area for which you want to create a boundary. You can continue picking additional points until you press Enter to end the command.
Alternate options: [Advanced/Undo]

7.160.3 Options within the -BOUNDARY command

After you begin creating boundaries, the following options may be available:

Advanced
Choose to modify boundary options or island detection.

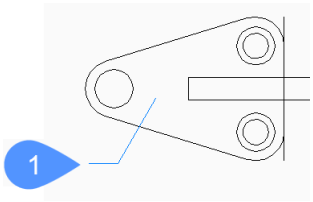
Boundary
Specify where BricsCAD should search for entities that make up the boundary.

- **New**: create a new selection set of entities that make up the boundary.
- **Everything**: select all entities in the current viewport.

Island
Specify the island detection. An island is an enclosed area inside a boundary.

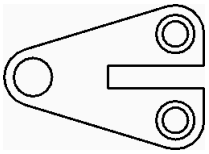
- **Yes**: treat every island as a boundary.
- **No**: create a boundary of only the outermost entities.
- **Outer only**: ignore the innermost areas. A boundary is created between the outermost area and islands.

Original entities:

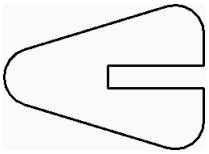


1 Internal point

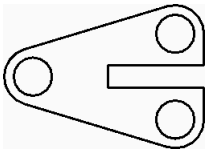
Yes:



No:



Outer only:



Exit
Return to the previous prompt.

Undo
Undo the last internal pick point and continue picking points to specify additional enclosed areas.

7.161 BOX command

Creates a 3D solid in the shape of a box.

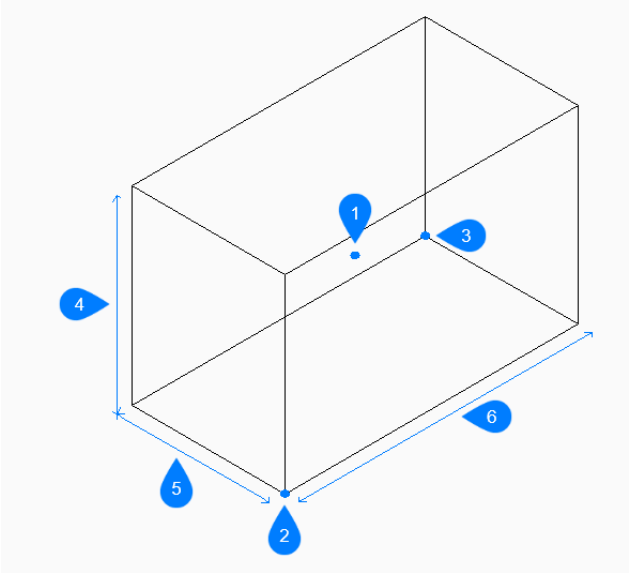
Note: In BricsCAD Lite, which doesn't support 3D Solids, the BOX command launches the AI_BOX command.

Shape Lite Pro Mechanical BIM

Icon:

7.161.1 Description

Create a 3D solid in the shape of a rectangular or square box. Choose from a combination of options including corner, center, length, width, height, and cube.



1 Center of box

2 Corner of box

3 Opposite corner

4 Height

5 Width

6 Length

7.161.2 Method

This command has 2 methods to begin creating a box:

- Set corner of box
- Center

7.161.3 Options within the command

Set corner of box
Allows you to specify a corner for the base of the box.

Set opposite corner
Allows you to specify the opposite corner for the base of the box to apply both the length and width. The box is created parallel to the X and Y axes.

Height of box
Specifies the height of the box.

Center
Allows you to begin creating a box by specifying the center of the box.

Cube
Allows you to specify a single distance to use for the length, width, and height of the box.

Length of side
Allows you to specify the length of the side of the box.

Width of box
Specifies the width of the box.

2Point
Specifies the height of the box by picking any two points.

7.162 BREAK command

Removes a portion of an entity.

Shape Lite Pro Mechanical BIM

Icon:

Alias: BR

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines.

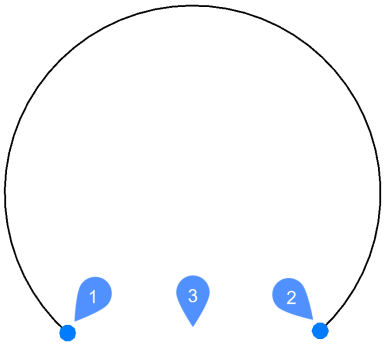
Note: Breaking a circle converts it to an arc. A ray is broken into a ray and a line. An infinite line is broken into two rays.

7.162.1 Method

When breaking entities, you must specify two points for the break. The portion between the two break points is removed.

On arcs and circles, the break occurs in the counter-clockwise direction from first to second break points.

Note: By default, the point you use to select the entity becomes the first break point.



1 First break point

2 Second break point

3 Removed portion

7.162.2 Options within the command

First break point
Specifies the start of the portion of the entity to remove.

same as first point (@)
Specifies that the first and second break points are at the same location on the entity. The selected entity is broken into two connected parts.

7.163 BREAKLINE command (Express Tools)

Creates a polyline with a breakline symbol.

Shape Lite Pro Mechanical BIM

Icon:

7.163.1 Method

Specify the first and the second point for the breakline, then specify the location for the breakline symbol.

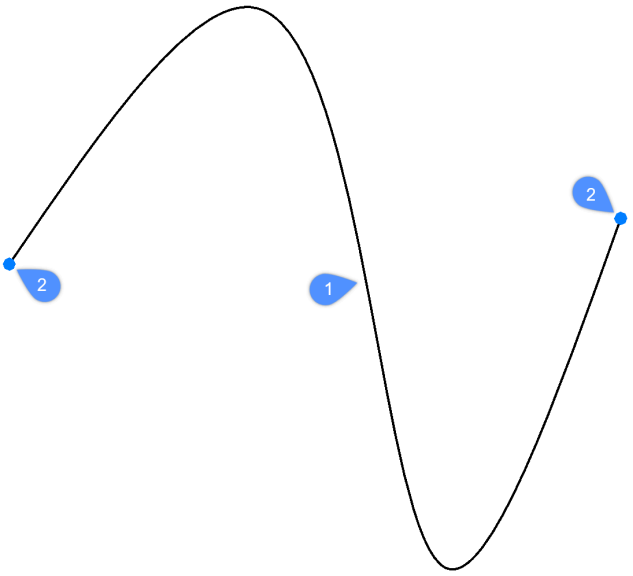
You can customize the breakline symbol with these steps:

1 Open a new drawing.

2 Draw a symbol (1).

3 Set the **Defpoints** layer current.

4 Add two points (2) to the symbol, using the POINT command. The breakline will intersect the symbol in these points.



5 Save this drawing in the ExpressTools folder. The default path is: *C:\Program Files\Bricsys\BricsCAD V25 en_US\ExpressTools*.

7.163.2 Options within the command

Block
Specifies the block that is used as a breakline symbol.

Note: The default block is defined by the **brkline.dwg**.

Size
Sets the size of the breakline symbol.

Note: The size of the breakline symbol determines the minimal length of the breakline.

Extend
Sets the extension length of the breakline when a break symbol is added.

7.164 BROWSER command

Opens the default Web browser.

Shape Lite Pro Mechanical BIM

7.164.1 Description

Opens the default Web browser to browse the internet starting from a specified URL. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

7.165 BSAVEAS command

Opens the **Save block definition** dialog box.

Shape Lite Pro Mechanical BIM

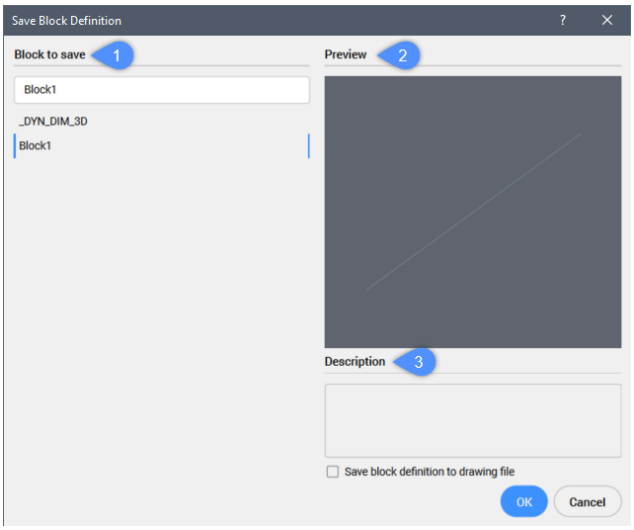
Icon: 

7.165.1 Description

Opens the **Save Block Definition** dialog box to copy the current block definition to a new name.

Note:

- This command is only available from the **Block Editor** and can be accessed from the Ribbon or called at the Command line.
- You can access the **Block Editor** with the BEDIT command or by double-clicking on the block you wish to edit.



1 Block to save

2 Preview

3 Description

7.165.2 Block to save

Specifies a new name for a selected block to make a copy of.

7.165.3 Preview

Displays a preview of the selected block.

7.165.4 Description

Displays the description of the selected block, if it has one.

7.166 BTRIM command (Express Tools)

Trims entities to entities nested in blocks and external references.

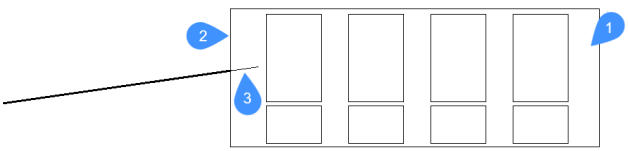
Shape Lite Pro Mechanical BIM

Icon:

7.166.1 Method

Select one or more entities (2) nested in blocks or external references (1) to use as cutting edge, then press ENTER.

Select the entity to be trimmed (3), and press ENTER.



1 Block/External reference entity

2 Cutting edge (an entity nested in the block/external reference entity, in this case a rectangle)

3 Entity to be trimmed

Note: Switch to extend mode: press and hold the Shift key to select an entity to be extended to the nearest boundary entities. See the BEXTEND command.

7.166.2 Options within the command

Fence
Selects all entities that cross the selection fence. The selection fence is a series of temporary line segments. The selection fence does not form a closed loop.

Crossing
Selects entities within and crossing a rectangular area defined by two points.

Edge mode
Toggles between **Extend** and **No extend**.

Extend
Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.

No extend
Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.

Projection
Specifies the projection method used when extending objects.

No projection
Extends only those entities that intersect real boundaries in 3D space.

xy plane of Ucs
Projects entities and boundaries to the x,y plane of the current UCS, and then extends the projected entities that would intersect projected boundaries.

current View
Projects entities into the current view, and then extends them accordingly.

eRase
Deletes selected entities.

7.167 BURST command (Express Tools)

Explodes blocks, converting the attributes values into text entities.

Shape Lite Pro Mechanical BIM

Icon: 

7.167.1 Method

Select the block that you want to explode and press Enter.

8. C

8.1 CAEANALYZE2D command

Runs a 2D Finite Element Analysis.

Shape | Lite | Pro | Mechanical | BIM

Icon:

8.1.1 Description

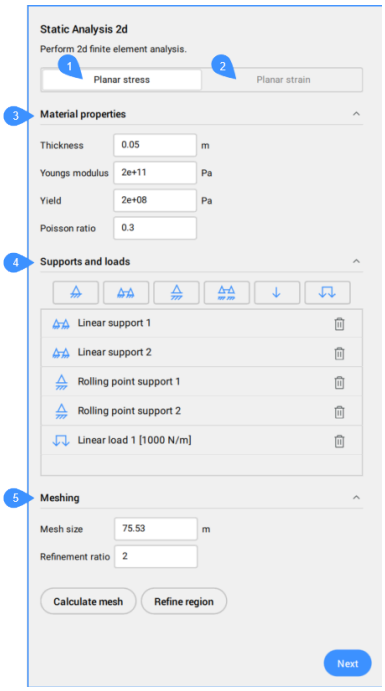
Runs a 2D Finite Element Analysis for a closed 2D boundary or 3D planar face that must be in the XY plane of the World Coordinate System.

8.1.2 Method

Select a closed 2D boundary or 3D planar face as input to open the **Static Analysis 2D** command context panel that allows you to define the material and physical parameters, loads and supports, and a mesh for the input boundary. Then run the analysis and visually inspect and interact with the analysis results.

The material and physical parameters include Young's modulus, the Poisson ratio, the yield, and the thickness. Loads include point forces and linear pressures, and supports include point and linear supports. The analysis results contain a summary of the input values, some global output values (execution time, maximum von Mises stress, maximum displacement, and a safety factor), and a colored mesh visualization of the displacement or the Von Mises stresses.

Note: The options within the Command line reflect the options found in the command context panel.



1 Planar stress mode

2 Planar strain mode

3 Material properties

4 Supports and loads

5 Meshing

Planar stress mode
Runs the analysis in Planar stress mode. This mode should be used when the stress tensor is assumed to not have out of plane components.

Planar strain mode
Runs the analysis in planar strain mode. This mode should be used when the strain in the Z-direction is assumed to be zero.

Material properties
Defines the material properties.

Thickness
Defines the thickness of the material.

Youngs modulus
Defines the Young modulus of the material.

Yield
Defines the Yield of the material.

Poisson ratio
Defines the Poisson ratio.

Supports and loads
Defines the Loads (point forces and linear pressures) and the supports (point and linear supports, rolling point and linear supports). Loads and supports can be placed over curved entities.

Point support
Defines the point support by clicking the point location on the boundary of the region in the drawing. The point support has a graphic representation (symbol) in the drawing.

Note: The annotation scale determines the size of the supports and loads symbols in the Model Space.

Linear support
Defines the linear support by clicking the first and second point locations on the boundary of the region in the drawing. The linear support has a graphic representation in the drawing.

Rolling point support
Defines the rolling point support by clicking the point location on the boundary of the region in the drawing. The point support has a graphic representation (symbol) in the drawing.

Rolling linear support
Defines the rolling linear support by clicking the first and second point locations on the boundary of the region in the drawing. The linear support has a graphic representation in the drawing.

Point load
Defines the point load by clicking the point location on the boundary of the region in the drawing. Then, specify the angle of the load to the tangent of the boundary (90 degrees by default, for example, perpendicular to the boundary) and specify the point load magnitude. The point load has a graphic representation in the drawing.

Linear load

Defines a linear load by clicking the first and second point locations on the boundary of the region in the drawing. Then, you have to specify the angle of the linear pressure to the tangent of the boundary (90 degrees by default, for example, perpendicular to the boundary) and to specify the linear pressure magnitude in. The linear load has a graphic representation in the drawing.

Graphic representation of the supports and loads:



List of supports and loads

Lists all added supports and loads. Each of them can be deleted.

Meshing

Defines a mesh for the input boundary.

Mesh size

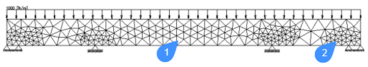
Defines the desired edge length of mesh elements.

Refinement ratio

Sets a refinement ration.

Calculate/update mesh (1)

Generates the mesh for the given region, mesh size, loads, and constraints. The mesh is shown on the selected region in the drawing. The mesh needs to be updated after changing the mesh size, loads, or constraints.

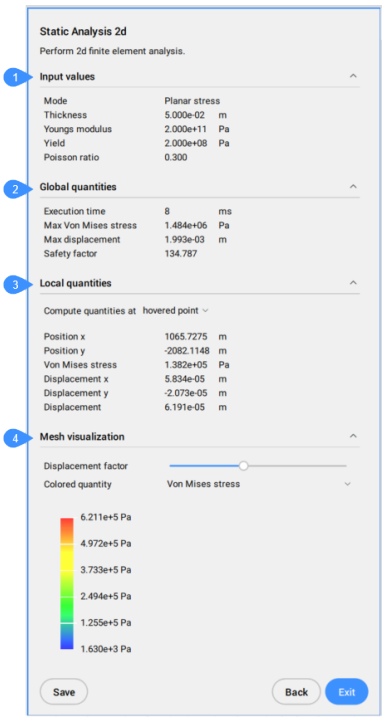


Refine region (2)

Adds a refined region where the generated mesh will be improved based on the specified refinement ratio.

Note: Locally refined areas will be lost when you update the entire mesh.

By pressing **Next** button, the result of analysis opens:



1 Input values

2 Global quantities

3 Local quantities

4 Mesh visualization

Input values
Displays the input values defined in the previous panel.

Global quantities
Displays the calculated output values.

Execution time
Displays the execution time.

Max Von Mises stress
Displays the maximum Von Mises stress.

Max, displacement
Displays the maximum calculated displacement.

Safety factor
Displays the calculated safety factor.

Local quantities
Displays the local output values on a specified point. Choose one option from the drop down list, **last clicked point** or **hovered point**.

Position x
Displays the X position of the point.

Position y
Displays the Y position of the point.

Von Mises stress
Displays the Von Mises stress in the point.

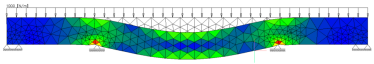
Displacement x
Displays the displacement of the point in x direction.

Displacement y
Displays the displacement of the point in y direction.

Displacement
Displays the total displacement of the point.

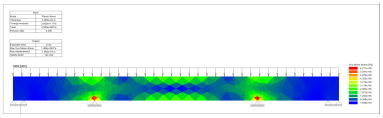
Mesh visualization
Controls the visualization of the results on the mesh.

Displacement factor
Defines the displacement factor.
The actual displacement of the nodes in the mesh is typically very small and hard to notice in the drawing. The displacement factor can be used to exaggerate the displacement of the nodes in the mesh.



Colored quantity
Displays a colormap legend with numerical values for **Displacement** and **Von Mises stress** (see the drop down list).

Save
Places the analysis results in the Model space.



8.2 CAL command

Opens the BricsCAD Calculator.

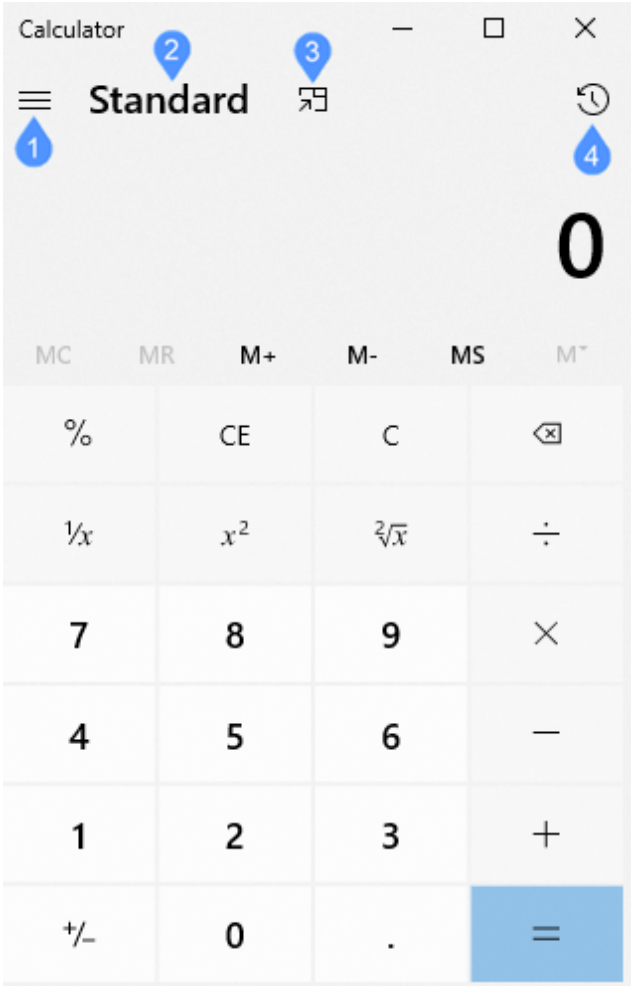
Shape Lite Pro Mechanical BIM

8.2.1 Description

Opens the BricsCAD Calculator to perform common calculation and conversion operations. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

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

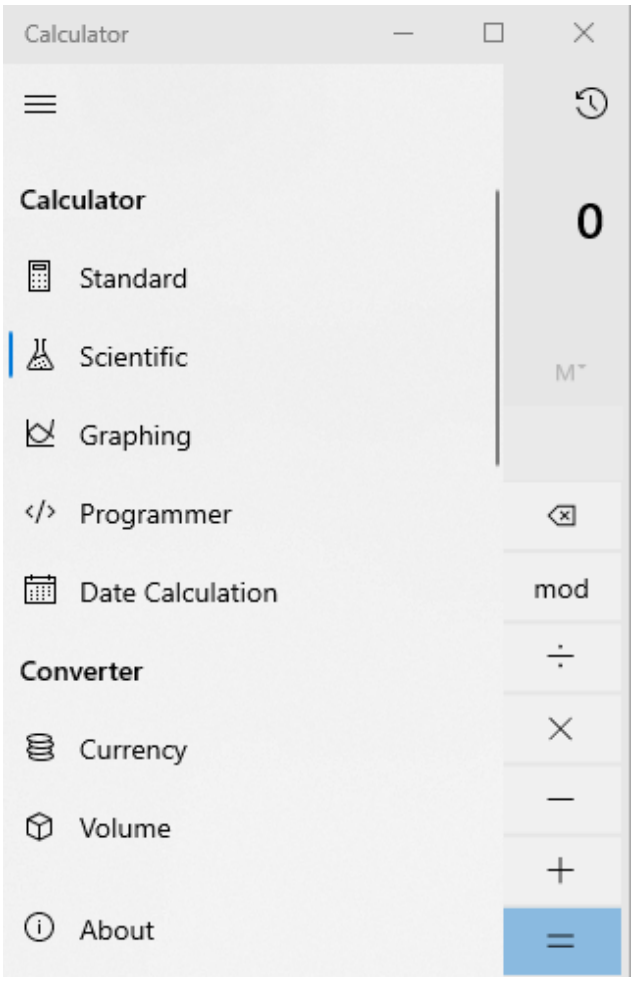
3 Keep on top

4 History

8.2.2 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).



8.2.3 View's name

Displays the current view name.

8.2.4 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.

8.2.5 History

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

Note: This function is available only for the Standard and Scientific calculator.

8.3 CALLOUT command

Shape Lite Pro Mechanical BIM

8.3.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

8.4 CAMERA command

Places camera glyphs in drawings that point at target points, and creates named views.

Shape Lite Pro Mechanical BIM

Icon:

8.4.1 Method

Specify the location of the camera and the location of the target (the point at which the camera looks).

A camera glyph displays in the drawing to indicate the position of the camera.

Note: A camera glyph displays in the drawing to indicate the position of the camera.

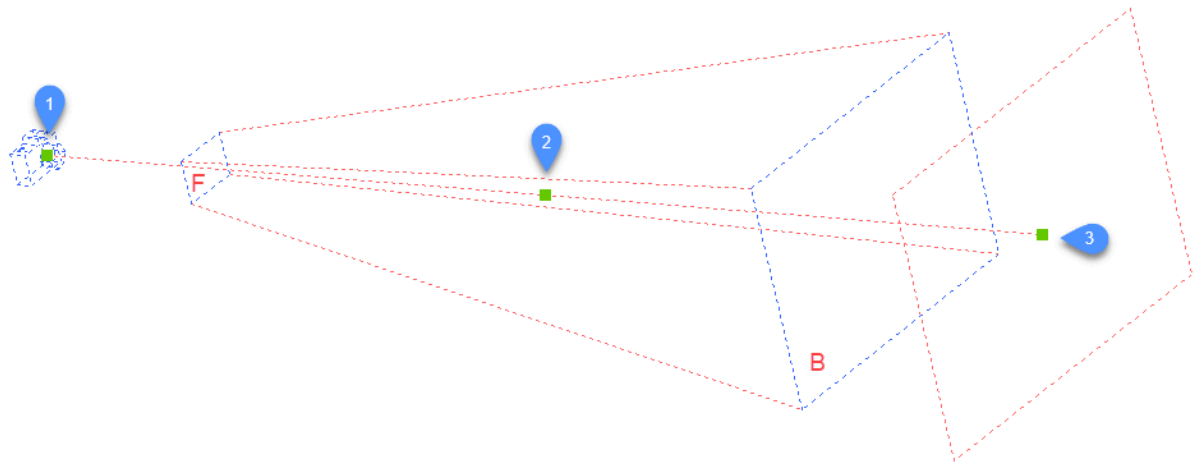
Grips Editing

Cameras can be edited directly through grips:

Select the camera glyph in the drawing. Notice that it has three grips. Front (F) and back (B) clipping planes display in a blue dotted line.

Drag the grips to perform the following actions:

- Position the camera (1).
- Move the camera definition as a whole (2).
- Position the target (3).



8.4.2 Options within the command

?

Displays a list of existing cameras. Press Enter to list all existing cameras.

Use wildcards (* or ?) to list a selection of cameras. For example, Cam* lists all cameras names that start with 'cam' and ?a* lists all camera names of which the second letter is 'a'

Name
Names the new camera.

LOcation
Positions the camera by picking a point in the drawing or typing the x, y, z coordinates in the Command line.

Height
Sets the height (z coordinate) of the camera.

Target
Positions the target, the point at which the camera looks by picking a point in the drawing or typing the x, y, z coordinates in the command line.

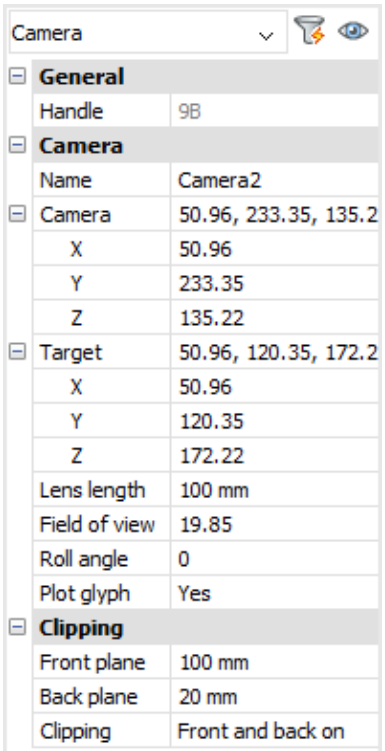
LEns
Defines the lens length. A smaller number, such as 20, provides a wider field of view, while a higher number, such as 200, provides a closer look, like a zoom lens on a camera.

CLipping
Defines the front and back clipping planes, which cut off the view.

View
Sets the camera as current view.

Note: Properties of a camera view can be edited:

- in the dialog box displayed by the VIEW command.
- in the Properties panel after selecting the camera glyph in the drawing.
-



8.5 CDORDER command (Express Tools)

Arranges the entities draw order based on their index color.

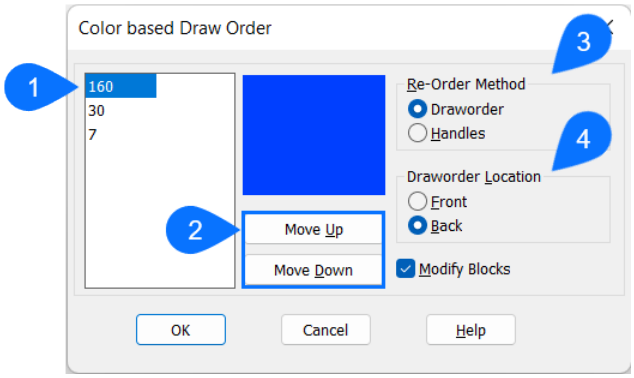
Shape Lite Pro Mechanical BIM

Icon:

8.5.1 Method

Select the entities you want to arrange. The **Color based Draw Order** dialog box opens, where you can specify the color priority.

The **Color based Draw Order** dialog box allows you to specify the draw order of the selected entities based on their index color.

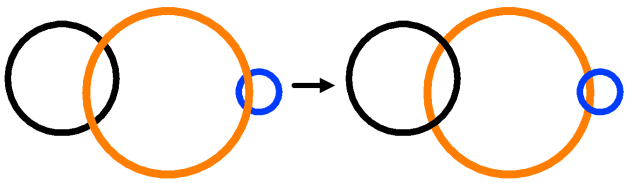


1 Color list

2 Move up and down buttons

3 Re-Order Method

4 Draworder Location



8.5.2 Color list

Lists the index colors of the selected entities.

Note: The first color on the list places the entities in the front, while the last color places the entities in the back.

8.5.3 Move up and down buttons

Move up

Moves the selected color up in the list.

Move down
Moves the selected color down in the list.

8.5.4 Re-Order Method

Draworder
Changes the display order of the selected entities using the DRAWORDER command.

Handles
Changes the display order of the selected entities by reordering entities in the drawing database.

8.5.5 Draworder Location

Front
Places the selected entities in front of the entities that are not specified in the color list.

Back
Places the selected entities behind the entities that are not specified in the color list.

Modify Blocks
Changes the display order of the entities in blocks.

8.6 -CDORDER command (Express Tools)

Arranges the entities draw order based on their index color via the command line.

Shape Lite Pro Mechanical BIM

8.6.1 Options within the command

Draworder
Changes the display order of the selected entities using the DRAWORDER command.

Enter one or more color number(s), comma-separated
Allows you to specify the colors indexes, separated by comma.

Note: The first color places the entities in the front, while the last color places the entities in the back.

Specify draworder location for entities
Allows you to choose between **Front** and **Back**.

Front
Places the selected entities in front of the entities that are not specified in the color list.

Back
Places the selected entities behind the entities that are not specified in the color list.

Redefine draworder inside selected blocks?
Changes the display order of the entities in blocks.

Y
The selected blocks are reordered.

N
The selected blocks remain unchanged.

Handles
Changes the display order of the selected entities by reordering entities in the drawing database.

8.7 CENTER command

Toggles the **Center** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



8.7.1 Description

Toggles the **Center** entity snap to enable or disable snapping to center. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

8.8 CENTERDISASSOCIATE command

Breaks the association of a centerline entity with two selected lines, or else a center mark with a circle or an arc.

Shape Lite Pro Mechanical BIM

Icon: 

8.8.1 Description

Breaks the association of centerline and center mark entities with the associated entities, such as lines, arcs, and circles.

8.9 CENTERLINE command

Creates a centerline.

Shape Lite Pro Mechanical BIM

Icon:

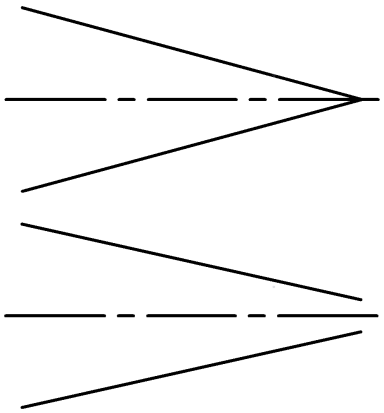


Alias: CL

8.9.1 Description

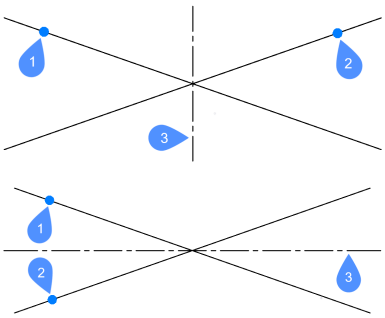
Creates a centerline geometry associated with two selected lines or polyline segments.

Note: Centerlines are drawing reference entities to indicate axes of symmetry.



8.9.2 Method

When applying a centerline to the intersection of two lines, the location of the picked points determines the direction of the centerline.



1 First segment selected.

2 Second segment selected.

3 Resulting centerline.

The lines and polyline segments can be selected inside blocks and in drawing view viewports. You can pick two segments of the same polyline.

Note: The centerline is associative, so when one or both lines are moved, the centerline repositions itself.

Note: Centerlines can be edited through grips. It can be moved and lengthened by dragging the grips. The CENTERRESET command can be used to reset the centerline.

8.10 CENTERMARK command

Creates a center mark.

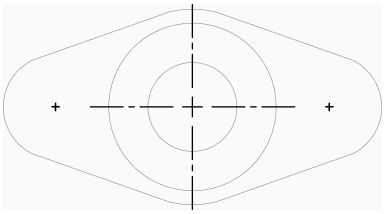
Shape Lite Pro Mechanical BIM

Icon:

Alias: CM

8.10.1 Description

Creates a center mark associated with a selected circle, arc, or polyarc.



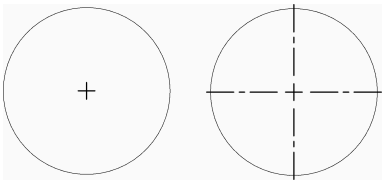
8.10.2 Method

There is one method to begin creating a center mark:

- Select circle or arc

Select circle or arc

Begin creating a center mark by selecting a circle, arc or polyarc. Depending on the current value of the CENTERMARKEXE system variable, the center mark is drawn with or without extension lines. Additional system variables further control the appearance of centermarks.



The center mark is associative, so when the arc or circle is moved or resized, the centermark follows.

8.11 CENTERREASSOCIATE command

Associates a centerline entity with two selected lines, or else a center mark with a circle or an arc.

Shape Lite Pro Mechanical BIM

Icon:

8.11.1 Description

Selects the center mark or a centerline to associate with their specific entities.

If you select a center mark you have to specify a circle or an arc (circular entity) to associate with.

If you select a centerline you have to specify two lines to associate with.

8.12 CENTERRESET command

Resets centerline and center mark entities.

Shape Lite Pro Mechanical BIM

Icon:

8.12.1 Description

Centerlines are reset by the program when one of the associated lines is moved. Center marks are reset automatically when the associated circle or arc is moved or when their radius or diameter is modified.

8.13 CHAMFER command

Creates chamfers at intersections, defined by two lengths, or a length and an angle.

Shape Lite Pro Mechanical BIM

Icon:

Alias: CHA

8.13.1 Method

There are four methods to create chamfers:

- Create chamfer by specifying two distances.
- Create chamfer by length and angle.
- Create chamfers along a polyline.
- Create chamfers on a 3D solid edge (obsolete, replaced by DMCHAMFER command).

Note: The command cannot place a chamfer between two polylines, but it can place a chamfer between a line and a polyline, even a closed polyline.

Note: To connect two entities without creating a chamfer, hold the Shift key when selecting the second entity. This acts like a combined Trim-Extend command.

8.13.2 Options within the command

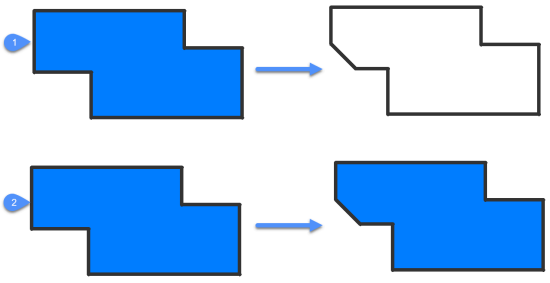
chamfer Settings

Opens the **Settings** dialog box at the chamfer parameters.

Polyline

Chamfers the vertices where two segments meet of the selected polyline (if applicable).

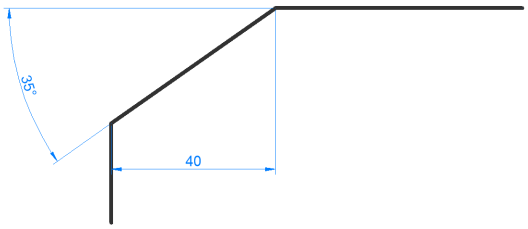
Note: Creating chamfers on a hatch boundary created with individual lines (1) results in removal of hatch associativity. Associativity is maintained if the boundary is defined from a polyline (2).



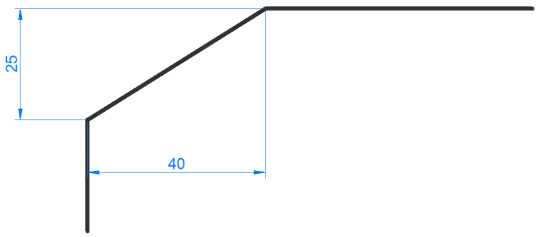
Angle

Changes the method to length-angle.

Note: BricsCAD measures the angle from 0 degrees, the X axis.



Distance
Changes the method to distance-distance.



mEthod
Specify between angle and distance methods.

Note: The program will keep using the same method for determining the chamfer until the method is changed again.

Trim
Determines if the entities are trimmed or extended to meet the endpoints of the chamfer line.

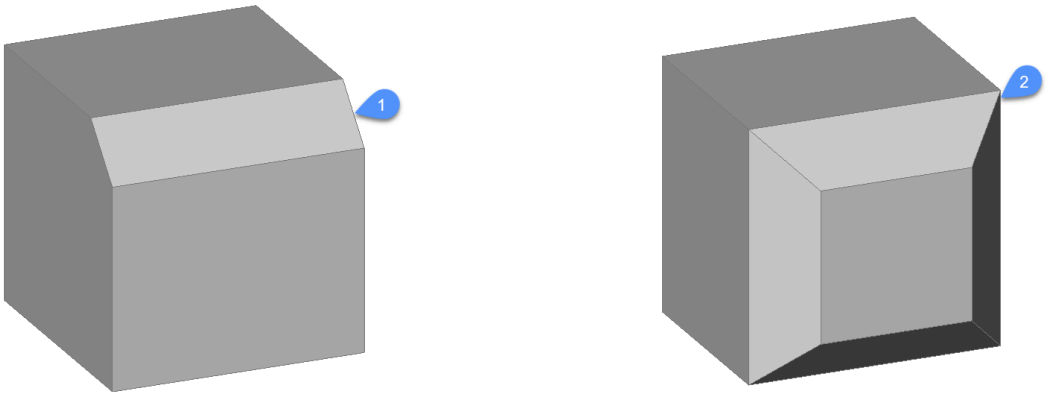
Undo
Undoes the last chamfer when in multiple mode.

Multiple
Create multiple chamfers with the same settings. Press the Esc key to exit the command.

Create 3D chamfer
Create 3D chamfer by selecting the edge of a 3D solid or surface.

Enter surface selection option
Select between surfaces adjacent to the selected edge.

Select edge or Loop
Select edge (1) to be chamfered or loop around selecting all tangential edges belonging to the base surface (2).



8.14 CHANGE command

Changes the properties of entities, through the Command line. This command has largely been superseded by the **Properties**panel.

Shape Lite Pro Mechanical BIM

Icon:

Alias: -CH

8.14.1 Method

Choose one or more entities in the drawing and choose a change point which changes the size of the entity, depending on what the entity is:

- Lines - moves the endpoint closest to the pick point.
- Arcs - changes the length closest to the pick point.
- Circles - changes the diameter.
- Text - moves the text.
- Blocks - moves the block.

Note: This option has no effect on polylines.

8.14.2 Options within the command

special Entities
Specifies options for changing the properties of rays and infinite lines (xlines).

change Angles
Changes the angle.

definition Points
Changes the location of the points defining the rays and infinite lines.

No changes
Leaves entities unchanged.

Properties…
Color/Elevation/LAyer/LineType/linetype Scale/Line Weight/ Thickness/TRansparency/Material/Annotative
For more explanation, see the **Properties** panel.

8.15 CHECKFORUPDATES command

Displays information about BricsCAD versions.

8.15.1 Description

Displays the current installed version and which is the latest available BricsCAD version for download. The download page will open in the browser when you click **Yes**.

8.16 CHECKSTANDARDS command

Checks the current drawing for inconsistencies that violate the standards.

Shape Lite Pro Mechanical BIM

Icon: 

8.16.1 Method

The CHECKSTANDARDS command is used to view and manage the violations of standards in the current drawing.

This command allows the user to fix or ignore each reported violation. The user can hide or show the ignored violations. If the ignored violations are hidden, they will no longer be reported as violations. See the STANDARDSOPTIONS system variable.

The user can choose to be notified about the standards violations. When the notification option is on, the user will be notified with an alert when a violation occurs. See the STANDARDSVIOLATION system variable.

The purpose of this command is to make some changes in the current drawing based on the standards that are defined for the drawing.

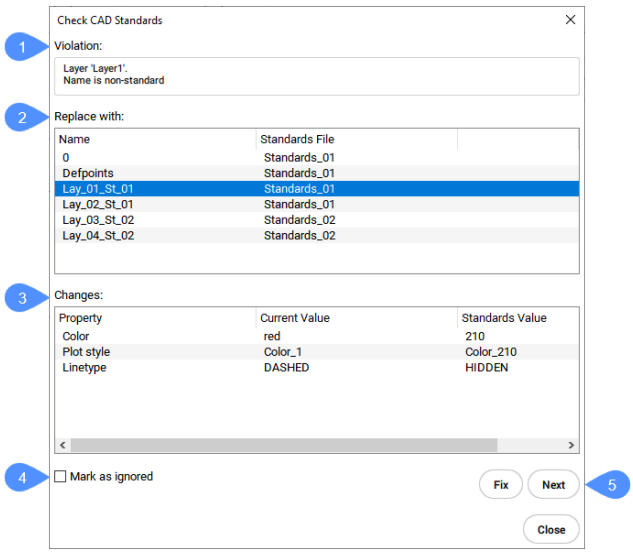
A standard is a set of properties of a named object in the standards file.

The properties of named objects in the current drawing are compared with the definitions in the standards file. When the named object does not comply with any standard, a violation occurs. In this case, the user can choose to apply a replacement. A replacement assigns a standard to a named object in the current drawing. All the properties of this named object are replaced by the values in the standard.

This command has an associated dialog box that manages all necessary operations.

8.16.2 CHECKSTANDARDS dialog box

The **Check CAD Standards** dialog box allows to select the options for the CHECKSTANDARDS command.



1 Violation

2 Replace with

3 Changes

4 Mark as ignored

5 Fix / Next / Close

Violation

This section contains a description of the current violation. The current violation includes a target named object in the current drawing that does not comply with the standards. The description includes the target named object type and name and the first property of the target named object.

Replace with

This section contains a list of standards that can be applied to the target named object. An entry of the list includes the name of the source named object in the standards, and the standards file name where this source named object is located. Clicking on an entry will highlight it and make it active.

Changes

This section displays the changes that can be made to the target named object. It contains a table that displays the properties of the target named object, in the current drawing. For each property, the table displays the current value and the standards value. The current value is the property value that is currently assigned to the target named object. The standards value is the property value defined in the standard. When the user chooses to make the change, the properties of target named object will take the values in the chosen standard.

Mark as ignored

This option allows to ignore a certain violation. The user can choose to hide the ignored violations. See the STANDARDSOPTIONS system variable.

Fix / Next / Close

This section contains the buttons assigned to the actions that can be taken.

Fix
Applies the selected standard to the target named object.

Next
Proceeds with the next violation in the current drawing.

Close
Closes the dialog box.

8.17 CHPROP command

Changes the properties of the selected entity.

Shape Lite Pro Mechanical BIM

8.17.1 Description

Changes the properties of the selected entities, with fewer options than the CHANGE command. You can change the properties for one or multiple entities.

Note: This command is largely superseded by the properties panel.

8.17.2 Options within the command

Color
Changes the color of the entity.

Truecolor
It allows you to specify the true color by introducing values for colors.

COlorbook
Select colorbook to append.

LAyer
Changes the layer to which the entities are assigned.

LineType
Changes the linetype. Linetype must be loaded into the drawing.

linetype Scale
Changes the linetype scale.

Line Weight
Changes the lineweight.

Thickness
Changes the thickness.

TRansparency
Changes the transparency.

Material
Applies a material.

Annotative
Sets the annotative property.

8.18 CHSPACE command

Moves entities from paper space to model space and vice versa.

Shape Lite Pro Mechanical BIM

Note: This command can only be used in a Paper Space.

8.18.1 Method

There are two methods to use CHSPACE command:

- Moves entities from paper space to model space.
- Moves entities from model space to paper space.

8.18.2 Options within the command

Note: The following options are available only with multiple viewports active

Select TARGET viewport

Select target viewport as reference.

Note: The scale factor is the inverse of custom scale of the selected target viewport. For example, if the custom scale = 1/2 (0.5), then the scale factor = 2.

Select SOURCE viewport

Select source viewport as reference.

Note: The scale factor equals the custom scale of the selected source viewport.

8.19 CHURLS command (Express Tools)

Changes previously placed URL addresses.

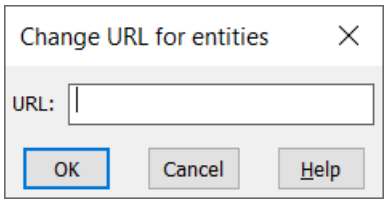
Shape Lite Pro Mechanical BIM

Icon:

8.19.1 Method

Opens the **Change URL for entities** dialog box.

The **Change URL for entities** dialog box allows you to change the URL previously attached to entities.



8.20 CIRCLE command

Creates circle entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: C

8.20.1 Description

Creates a circle from a combination of options including the center, radius, two points, three points or tangent to entities. You can also convert an arc to a circle.

8.20.2 Methods

There are 8 different methods to create circle entities:

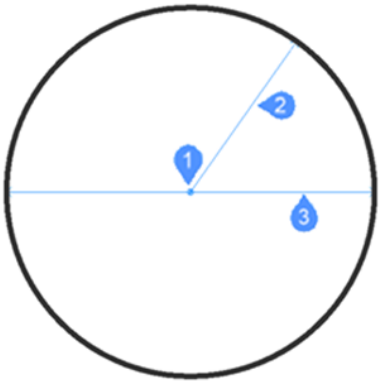
- Circle Center-Radius
- Circle Center-Diameter
- Circle 2-points
- Circle 3-point
- Circle Tangent-Tangent-Radius
- Circle Tangent-Tangent-Tangent
- Convert Arc to Circle
- Multiple circles

8.20.3 Options within the CIRCLE command

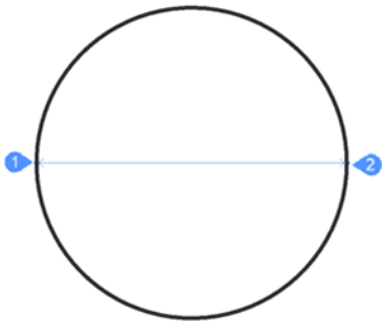
Select center of circle
Allows you to specify the center of the circle (1).

Set Radius
Allows you to specify the radius of the circle (2).

Set Diameter
Allows you to specify the diameter of the circle (3).

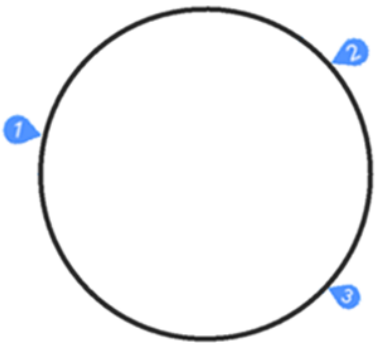


2 Point
Allows you to specify 2 diametrically opposite points (1 and 2) to define the circle.



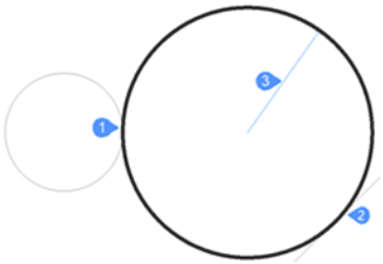
3 Point

Allows you to specify 3 points to define the circle.



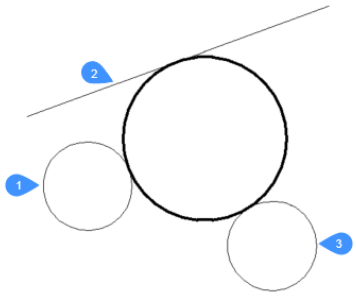
Tangent-Tangent-Radius

Defines the circle by two tangent points to other entities (1 and 2) and its radius (3). When you specify a radius that is not possible with the selected tangents, you are prompted to specify the second tangent and the radius again.

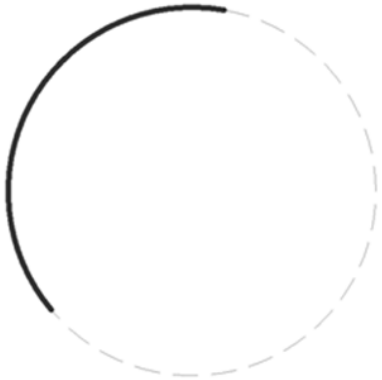


Tangent-Tangent-Tangent

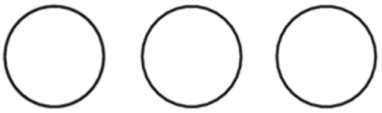
Defines the circle by three tangent points to other entities (1, 2, and 3).



turn Arc into circle
Converts an arc entity into a circle by extending it to 360 degrees.



Multiple circles
Creates multiple circles with the same size using one of the listed methods above. Press **Enter** to end the command.



8.21 CIVIL3DIMPORT command

Creates BricsCAD civil entities from an Autodesk® Civil 3D drawing.

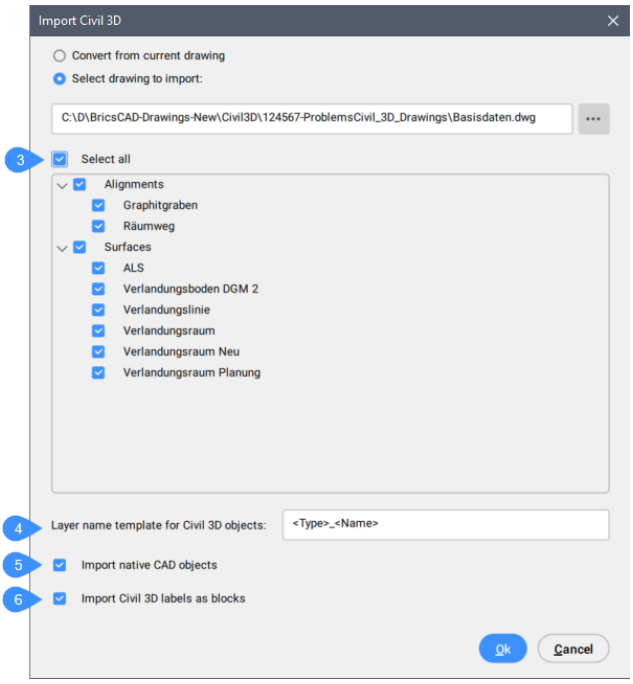
Shape Lite Pro Mechanical BIM

8.21.1 Description

Opens the **Import Civil 3D** dialog box which allows you to import AutoCAD® Civil 3D entities from a selected DWG file, or to convert them directly from the current drawing.

The following objects from AutoCAD® Civil 3D can be imported, which are in BricsCAD also created as Civil entities:

- Cogo points and point groups.
- TIN Surfaces.
- TIN Volume Surfaces.
- Grid Surfaces.
- Alignments (both Alignments by PI and Alignments by Elements are supported).
- Profile Views.
- Profiles (Vertical Alignments).



1 Convert from current drawing

2 Select drawing to import

3 Select All

4 Layer name template for Civil 3D objects

5 Import native CAD objects

6 Import Civil 3D labels as blocks

8.21.2 Convert from current drawing

Converts AutoCAD® Civil 3D objects in the current drawing.

8.21.3 Select drawing to import

Enables to enter drawing path or click the **Select drawing** button to select the drawing manually.

Note: When either the DWG file is specified or the **Convert From Current Drawing** option is selected, AutoCAD® Civil 3D objects are listed in a tree view, organized by object type.

8.21.4 Select All

Enables to select all AutoCAD® Civil 3D objects in the selected drawing.

8.21.5 Layer name template for Civil 3D objects

Enables to specify the Layer name template for imported AutoCAD® Civil 3D objects.

The following settings are available for defining the Layer name template:

- **<Type>**: includes AutoCAD® Civil 3D object type in the Layer name.

- **<Name>**: includes AutoCAD® Civil 3D object name in the Layer name.
- **<SourceLayer>**: Civil entities are created on same layer as source AutoCAD® Civil 3D objects.

If you want to draw BricsCAD Civil entities on Layers, which names will consist of three sets: the type of the source Civil 3D object, the object name and the name of the Layer of the source AutoCAD® Civil 3D object, you need to write the following text in the field:

```
<Type>_<Name>_<SourceLayer>
```

8.21.6 Import native CAD objects

Enables to import native CAD objects from the selected drawing in addition to AutoCAD® Civil 3D objects.

8.21.7 Import Civil 3D labels as blocks

Enables to import AutoCAD® Civil 3D labels as blocks from the selected drawing.

Note: AutoCAD® Civil 3D COGO points labels are converted to BricsCAD Civil point labels.

8.22 CIVILDWGEXPORT command

Exports BricsCAD Civil entities as native CAD entities.

Shape Lite Pro Mechanical BIM

Icon: DWG

8.22.1 Description

Opens the **Save Drawing As** dialog box which allows you to export Civil entities as native CAD entities in a new drawing. You can flatten all the entities to zero elevation while keeping the annotation of spot elevation or contour labels with the correct representation.

8.22.2 Options within the command

Flatten entities
Allows you to flatten all entities.

Yes
Flattens all the entities to zero elevation while keeping the annotation of spot elevation or contour labels in the correct representation.

No
Does not flatten the entities.

8.23 CIVILEXPLORERCLOSE command

Closes the **Civil Explorer** panel.

Shape Lite Pro Mechanical BIM

8.23.1 Description

Closes the **Civil Explorer** panel to hide it from the current workspace. If the **Civil Explorer** panel is stacked when you close it, the **Civil Explorer** tab or icon is removed from the stack.

8.24 CIVILEXPLOREROPEN command

Opens the **Civil Explorer** panel.

Shape Lite Pro Mechanical BIM

8.24.1 Description

Opens the **Civil Explorer** panel to display it in the current workspace. The **Civil Explorer** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Civil Explorer** panel can be either floating, docked or stacked.

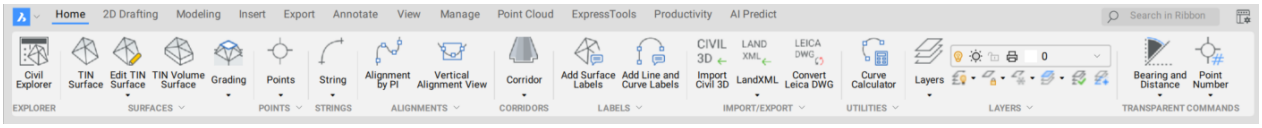
Civil Explorer is the main panel to manage and access Civil entities. Entities are arranged in a tree view and grouped by entity type.

Through this panel, you can access settings and properties of Civil entities and allows you to edit existing and add new entities and their components.

The panel is divided into two parts. The upper part with list of entities in a tree view, and the bottom part with additional properties, depending on the selection in the tree view.

The **Civil Explorer** panel is the core panel for Civil design. In the Civil Workspace, click the **Civil Explorer** icon in the Ribbon under the **Home** tab to toggle the panel on/off.

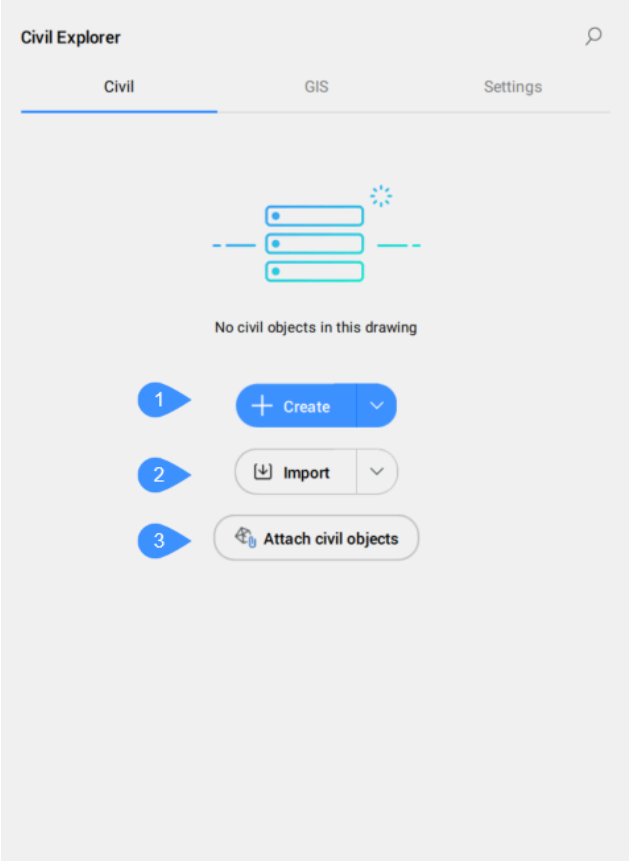
Note: Instead of individual Civil points, only point groups are listed on the **Civil Explorer**. Grading elements are listed in the tree-view, but it is not possible to edit their properties via **Civil Explorer**.



The **Civil Explorer** panel contains three tabs: **Civil**, **GIS** and **Settings**.

8.24.2 Civil Tab

If there are no Civil objects in the drawing, three buttons are available within the **Civil** tab.

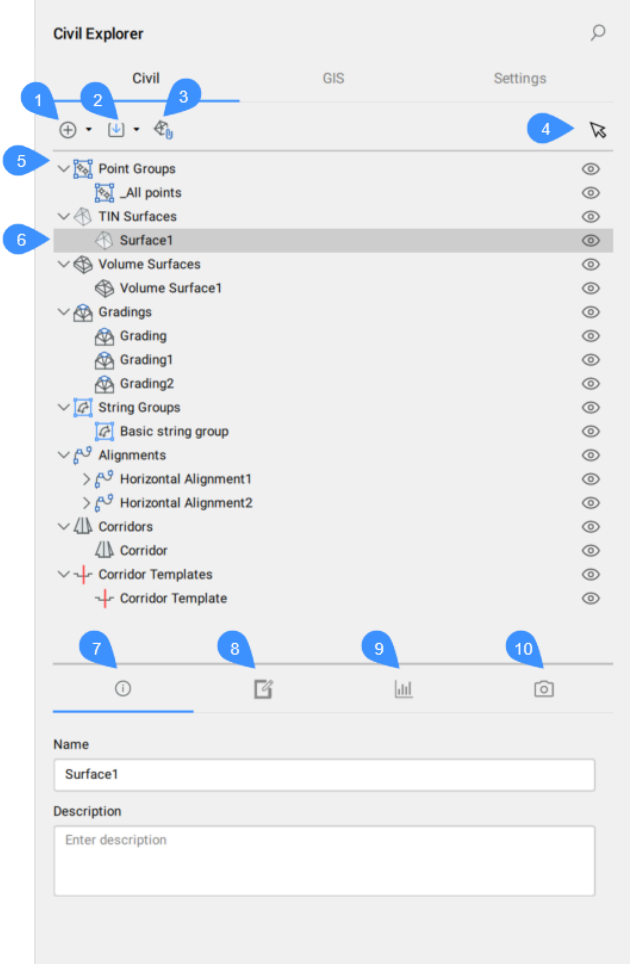


1 Create

2 Import

3 Attach civil objects

The **Civil** tab enables you to manage and access Civil entities in the current drawing.



1 Create

2 Import

3 Attach Civil objects

4 Pick a Civil object

5 Civil object types

6 Selected Civil object

7 Info

8 Definitions

9 Statistics

10 Visual Styles

Create

Allows you to create all types of the Civil objects. You have to select from the drop down list the Civil object type you want to create.

TIN Surface

Launches the TIN command.

Volume Surface
Launches the TINVOLUME command.

Grading
Launches the GRADING command.

Civil point
Launches the CIVILPOINT command.

Point Group
Launches the CIVILPOINTGROUP command.

String
Launches the STRING command.

Alignments
Launches the ALIGNMENT command.

Corridors
Creates corridors.

Corridor
Launches the CORRIDOR command.

Corridor Template
Launches the CORRIDORTEMPLATE command.

Template Element
Launches the CORRIDORTEMPLATEELEMENT command.

Import
Imports civil drawing files.

Import Civil 3D...
Launches the CIVIL3DIMPORT command.

Import LandXML...
Launches the LANDXMLIMPORT command.

Convert Leica DWG
Launches the LEICACONVERT command.

Attach civil objects
Launches the ATTACHCIVILOBJECT command.

Pick a Civil object
Pick a Civil object in the drawing to highlight it in the **Civil Explorer** panel.

Civil object types
Civil objects are arranged in a tree view. At the top of the tree structure, the types of objects that already exist in the drawing are listed. At the lower level, under each type of objects, the objects that already exist in the drawing are collected.
Right-click the Civil object type to open a context menu that contain the **Create** option, which launches specific commands to create a selected Civil object.
The right-click menu also contains a **Hide** option that can toggle the visibility for each type of Civil object.

Selected Civil object

Highlights the selected Civil object in a tree-view.

Right-click the entity to open a context menu that contains entity-specific **Edit** options, as well as options common to all entities, such as **Zoom to**, **Select**, **Delete**, **Pan to**, and **Hide**.

The available edit options depend on the entity type:

Points

Existing points are not listed on the Civil tab in the **Civil Explorer**. You can create new Civil Points by clicking the Points icon in the Ribbon under the Home tab.

Point Groups

Select the **Create** option from the context menu to open the **Point Group** dialog box.

The context menu for already defined Point Groups contains the **Civil Point Editor...**, **Edit point group...**, **Set styles...**, **Export points...**, **Zoom to**, **Select**, **Pan to**, **Delete**, and **Hide** options.

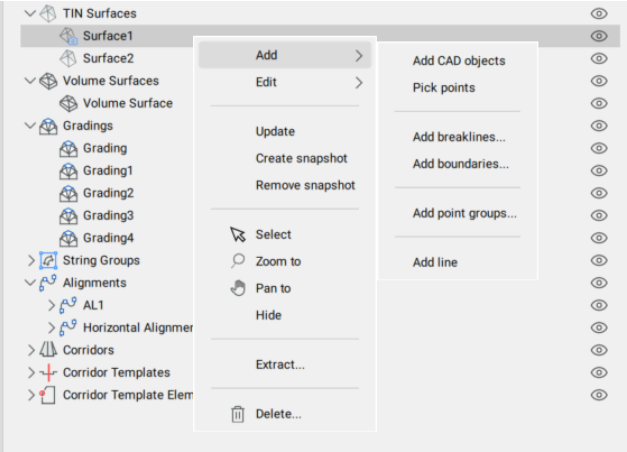
Note:

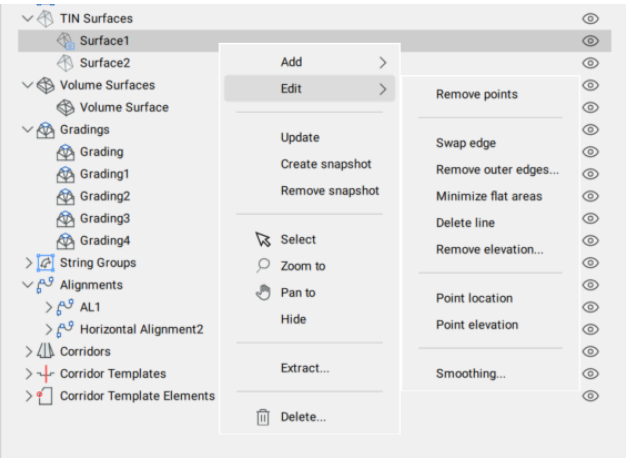
- Double-click a **Point Groups** node to open the **Civil Point Editor** dialog box.
- **Edit point group...** option opens the **Point Group** dialog box.
- **Set styles...** option opens the **Set Styles** dialog box to set the Symbol and Label styles for the point group.
- **Export points...** option opens the **Export Civil Points** dialog box.

TIN Surfaces

Using the **Civil Explorer**, you can **Edit** existing and **Add** new elements to existing TIN Surfaces.

Note: All the options under **Add** and **Edit** in the context menu are also available in the TIN and TINEDIT commands.





In addition to these options, you can **Update**, **Create snapshot**, **Remove snapshot**, or **Extract**.

Note: The **Extract...** option opens the **Extract from** dialog box.

Use the **Settings** tab in **Civil Explorer** panel to edit Surface label styles.
Also, TIN surfaces attached to the drawing with the ATTACHCIVILOBJECT command are listed.

Volume Surfaces

Update, **Select**, **Zoom to**, **Pan to**, **Hide**, **Extract**, and **Delete** options are available for Volume Surfaces.

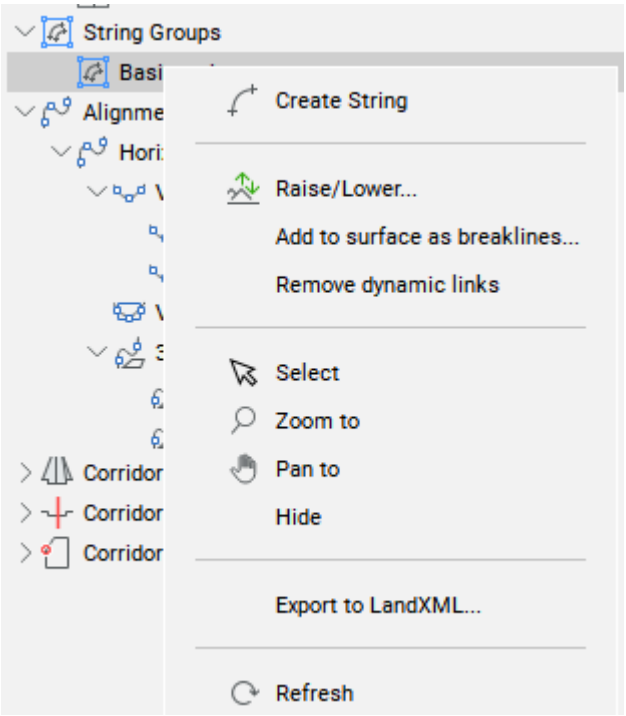
Gradings

In addition to the common options, there are **Split** and **Merge** options available for Gradings.

Note: These options are also available in the GRADINGEDIT command.

String Groups

Lists all String groups created. The context menu contains the option **Create string group**.
In addition to the common options, the context menu of a string group has the options:



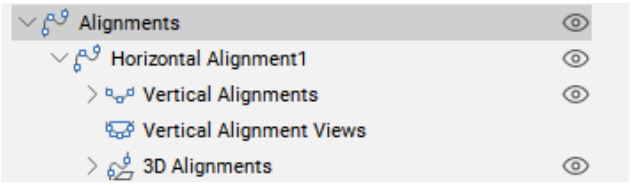
Create String
Launches the STRING command.

Raise/Lower
Allows you to raise or lower the elevation of Strings in selected String Group.

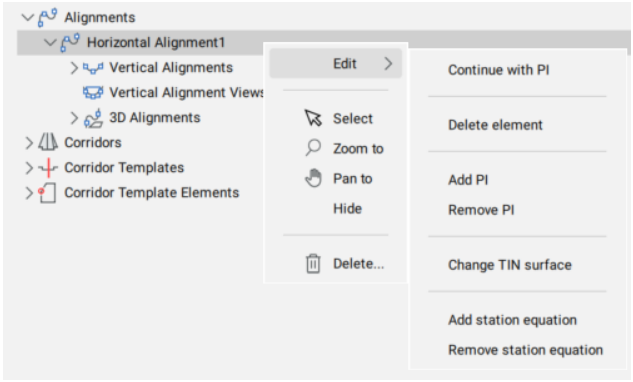
Add to surface as breaklines
Allows you to select the surface from the drop down menu, then opens **Add Breaklines** dialog box.

Export to LandXML
Allows you to export Strings in LandXML file.

Alignments
The Alignments entity type collects existing Horizontal Alignments and associated Vertical Alignments, 3D Alignments and Vertical Alignment Views.



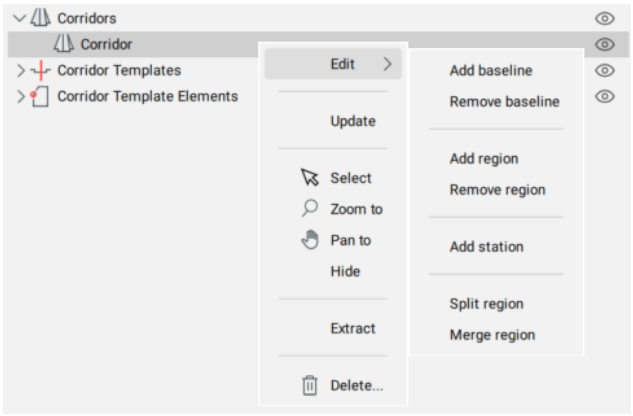
Right-click the Horizontal Alignment provides all the **Edit** options available in the ALIGNMENTEDIT command.



While for Vertical Alignments, Vertical Alignment views and 3D Alignments, the **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** options are available.

Corridors

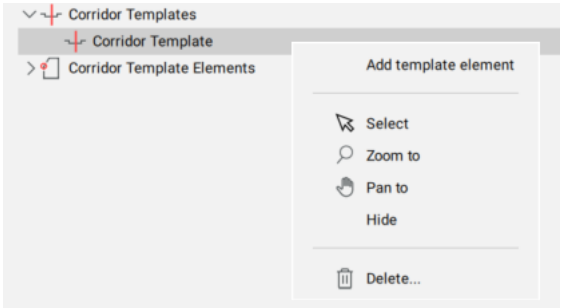
Right-click the existing Corridor provides all the **Edit** options available in the CORRIDOREDIT command. Besides **Edit** options, also the **Update, Select**, **Zoom to**, **Pan to**, **Hide**, **Extract**, and **Delete** options are available.



Corridor Templates

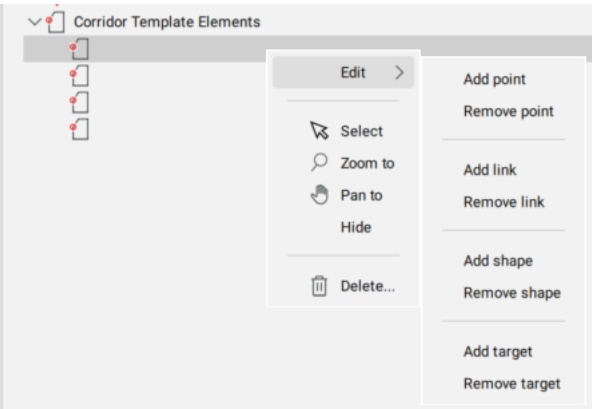
You can choose between **Add template element**, **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** options.

Note: The **Add template element** option is also available in the CORRIDORTEMPLATE command.



Corridor Template Elements

Right-click the Corridor Template Elements provides all the edit options available in the CORRIDORTEMPLATEELEMENTEDIT command. Besides **Edit** options, also the common options, such as **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** are available.



Info

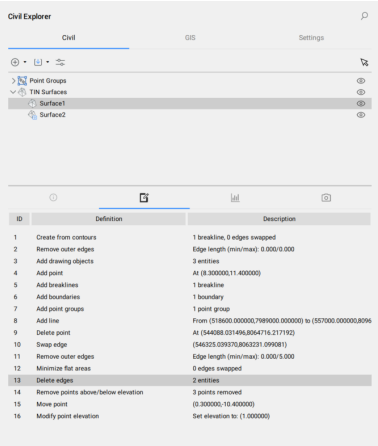
Displays information about the selected Civil object (Name and Description). Both can be edited.

Note: For String Groups, 'Interact with other strings in the group' switch is available.

Definitions

It shows the components (source data) from which the TIN surface was built, the editing operations used, and the chronological order in which definitions were added.

The surface build process for a TIN surface is incremental. When you add data to a surface or edit the surface, the surface is updated. When data is removed, the surface is rebuilt. The definitions list supports the incremental build process and sequentially shows all operations performed on the surface in its current state.

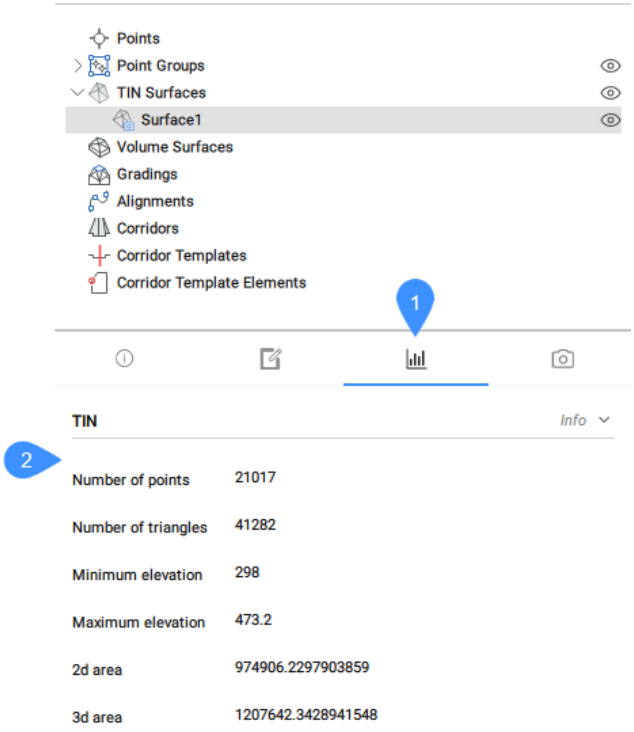


The ID field indicates in which step each component was added to the TIN Surface.

The context menu offers the possibility to change the order of definitions, to **Delete** or **Disable** them. Additionally, for most TIN Surface definitions, there are the options **Select** and **Zoom to**, which makes finding them in the drawing very easy. Double click a definition or press the **Edit** option from the context menu to open the **Surface: Surface name, Definition step ID** dialog box, which displays all details for the selected definition. You can edit the definition parameters.

Statistics

The **Statistics** tab displays the statistics for the selected TIN surface (number of points and triangles, minimum and maximum elevation, 2D and 3D area).



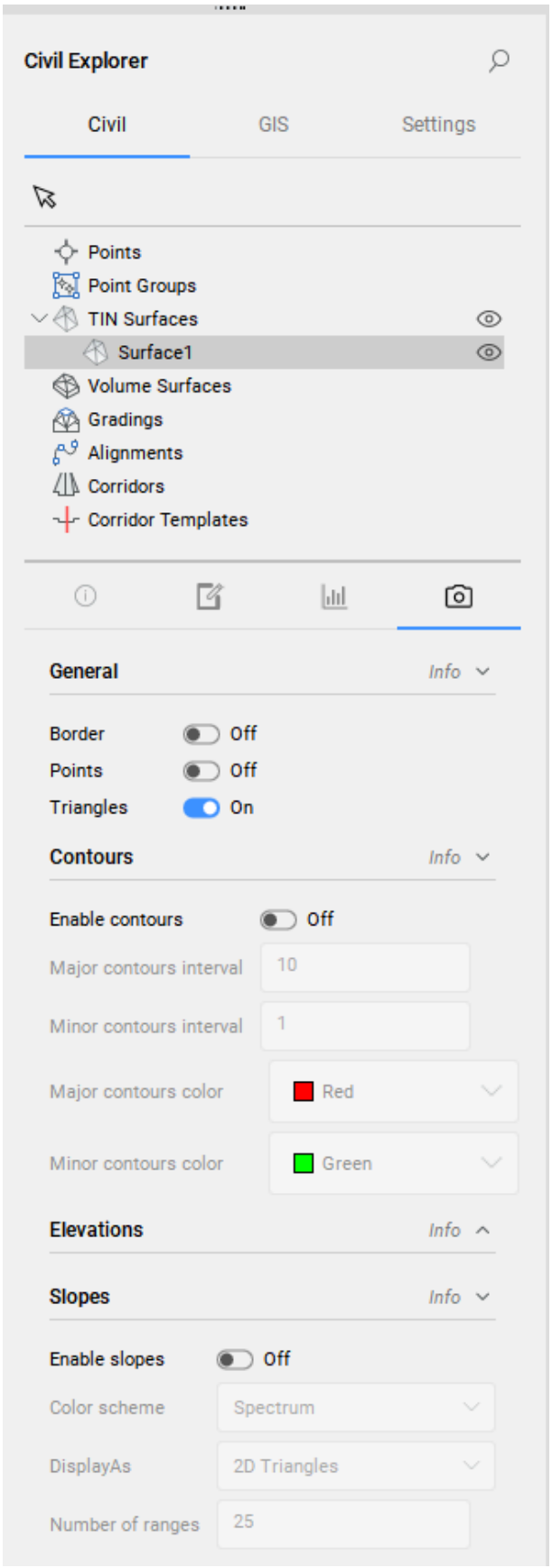
1 Statistics tab

2 Statistics data

Visual styles

The **Visual styles** tab allows you to set the visual styles for the TIN surface:

- general visual style – choose which components of the TIN surface to display (Border, Points, Triangles).
- visual style for contours.
- visual style for the elevation analysis.
- visual style for the slope analysis.


Civil Explorer
Civil
GIS
Settings
Points
Point Groups
TIN Surfaces
Surface1
Volume Surfaces
Gradings
Alignments
Corridors
Corridor Templates
General
Info
Border
Off
Points
Off
Triangles
On
Contours
Info
Enable contours
Off
Major contours interval
10
Minor contours interval
1
Major contours color
Red
Minor contours color
Green
Elevations
Info
Slopes
Info
Enable slopes
Off
Color scheme
Spectrum
DisplayAs
2D Triangles
Number of ranges
25

General
Enables/disables the display of certain TIN surface components.

Border
Toggles the display of the Boundary line of the TIN surface.

Points
Toggles the display of the Points of the TIN surface.

Triangles
Toggles the display of the Triangles of the TIN surface.

Contours
Specifies parameters for displaying contours on a TIN surface.

Enable Contours
Toggles the display of the Contours of the TIN surface. The visual style of the contours can be adjusted.

Major contours interval
Allows you to enter the interval for the major contours.

Minor contours interval
Allows you to enter the interval for the minor contours.

Major contours color
Allows you to enter the color for the major contours.

Minor contours color
Allows you to enter the color for the minor contours.

Elevations
Specifies parameters for creating TIN Elevations Analysis.

Enable Elevations
Toggles the display of the Elevations of the TIN surface. The visual style of the elevations can be adjusted.

Create ranges by
Allows you to choose between the methods of creating Elevation Analysis:

- Number of ranges
- Range interval
- Range interval with datum

Color scheme
Allows you to select the color scheme for the Elevation Analysis.
Choose between the available options:

- **Spectrum**
- **Earth**
- **Hydro**
- **Gray**
- **Reds**
- **Greens**
- **Blues**

Above datum color scheme
Allows you to select the Above datum color scheme.

Below datum color scheme
Select the Below datum color scheme.

DisplayAs
Select the TIN Surface elements used to create the Elevation Analysis.
Choose between the available options:

- **Triangles**
- **Contours**
- **Points**
- **2D Triangles**
- **2D Contours**

Range interval
Specifies the Range interval for the Elevation Analysis.

Number of ranges
Specifies the number of ranges for the Elevation Analysis.

Datum
Specifies the datum elevation.

Slopes
Specifies parameters for creating TIN Slopes Analysis.

Enable Slopes
Toggles the display of the Slopes of the TIN surface. The visual style of the slopes can be adjusted.

Color Scheme
Allows you to select the color scheme for the Slopes Analysis.
Choose between the available options:

- **Spectrum**
- **Earth**
- **Hydro**
- **Gray**
- **Reds**
- **Greens**

DisplayAs
Select the TIN Surface elements used to create the Slopes Analysis.
Choose between the available options:

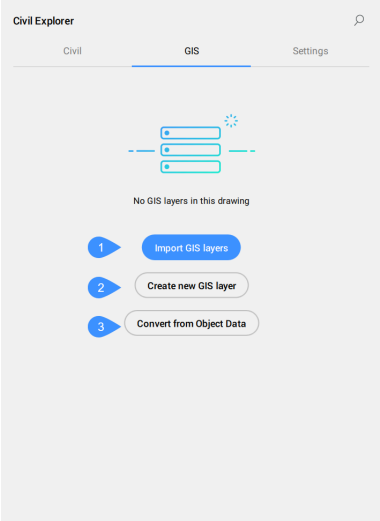
- **Triangles**
- **2D Triangles**

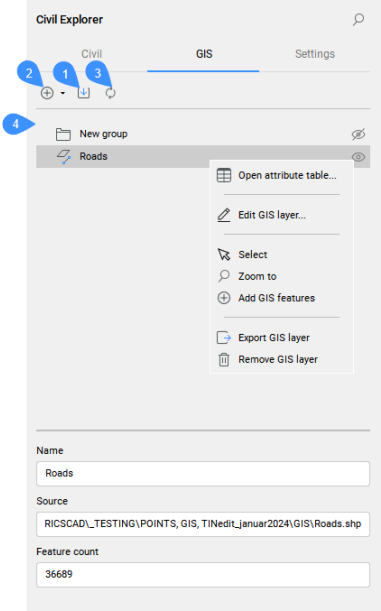
8.24.3 GIS Tab

GIS Tab contains a list of all the GIS layers in the drawing and helps you manage their visibility.

If there is no GIS layer in the drawing, there are three options on the GIS tab:

- Import GIS Layer
- Create GIS Layer
- Convert from Object Data





1 Import GIS layers

2 Create group and Create GIS layer

3 Convert from Object Data

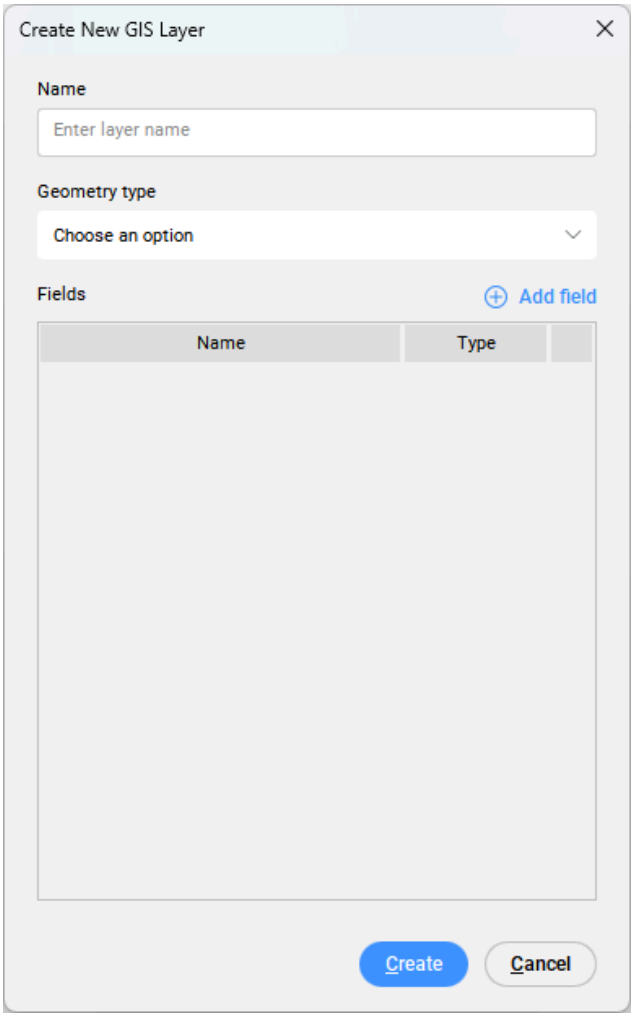
4 GIS tree view

Import GIS layers

Allows you to import ESRI SHP or ESRI Geodatabase files. Check the GISIMPORT command for the detailed description of **GIS Import** dialog box.

Create GIS layer

Allows you to create a New GIS Layer. The **Create New GIS Layer** dialog opens when the command is launched:



Name

Allows you to enter the new Layer name.

Geometry type

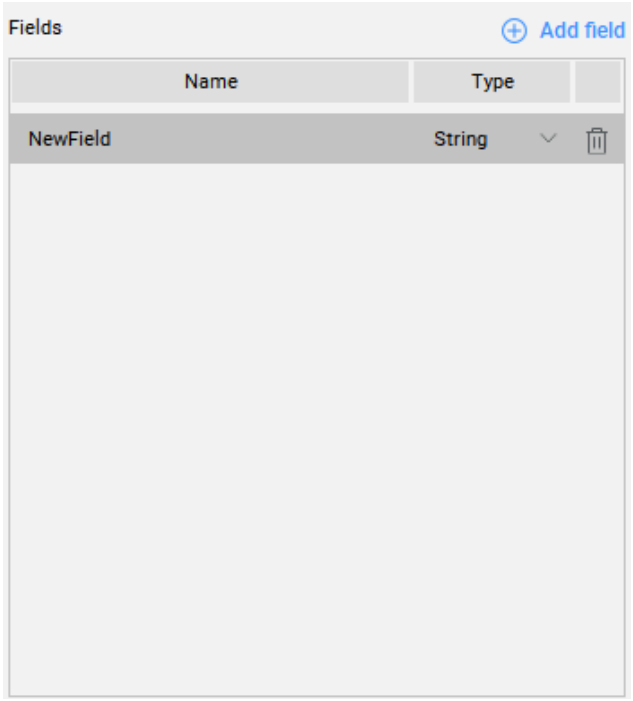
Allows you choose between the following geometry types:

- **Point**
- **Line**
- **Polygon**

Add Field

Adds a new field.

When clicking the **Add field** button, a new field is added to Fields list, with the default name **NewField** and default type **String**. You can edit the **Name** and **Type** fields.

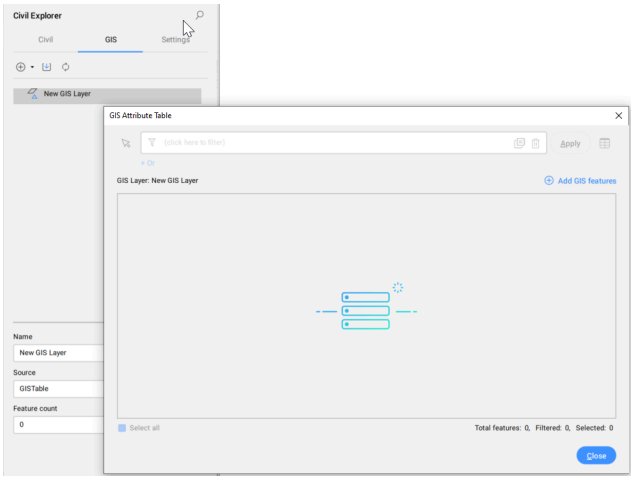


The following field Types are available:

- **String**
- **Real**
- **Integer**

Click the trash icon in the table to delete fields.

When you first open the table of a newly created GIS Layer, the table is empty. To add GIS features to the GIS Layer, click the button **Add GIS features**. Select linear entities in the drawing and press Enter. The **GIS Attribute table** dialog box contains selected GIS features with appropriate GIS data.



Convert from Object Data

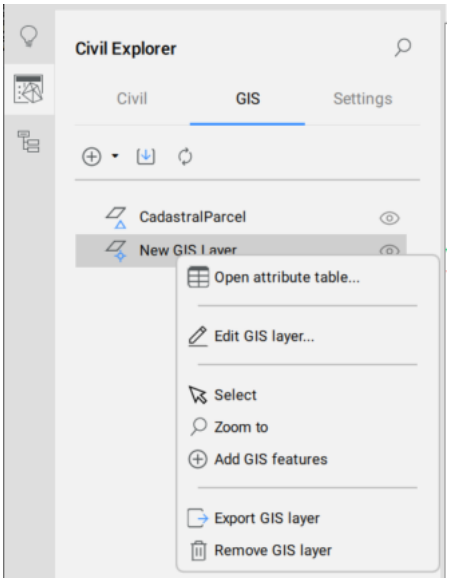
Converts AutoCAD® Map 3D and Autodesk® Civil 3D GIS features from current drawing to BricsCAD GIS data.

Note: The AutoCAD® Map 3D and Autodesk® Civil 3D Object Data can be viewed and edited directly in the **Properties** panel.

GIS tree view
Lists all GIS layers and groups.

Note: You can rearrange groups and GIS layers by using drag and drop.

The following context menu opens when you right-click a GIS Layer:



Open attribute table...
Opens the **GIS Attributes Table** dialog box.

Edit GIS layer
Opens **Edit GIS Layer** dialog box.

Select
Selects the GIS Layer.

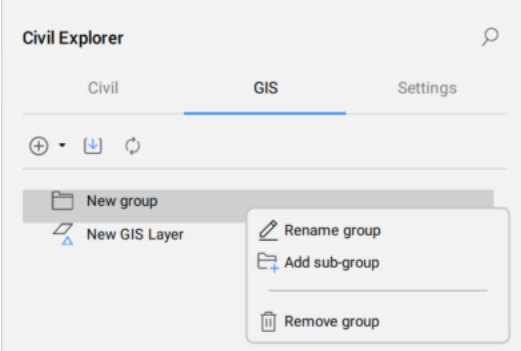
Zoom to
Zooms to the GIS Layer.

Add GIS features
Adds entities to a GIS Layer. A command prompt for selecting linear entities in the drawing is shown. The **GIS Attributes Table** dialog box opens where the new GIS features are added at the end of the table.

Export GIS layer
Exports the GIS layer.

Remove GIS layer
Removes the GIS layer.

The following context menu opens when you right-click a Group:



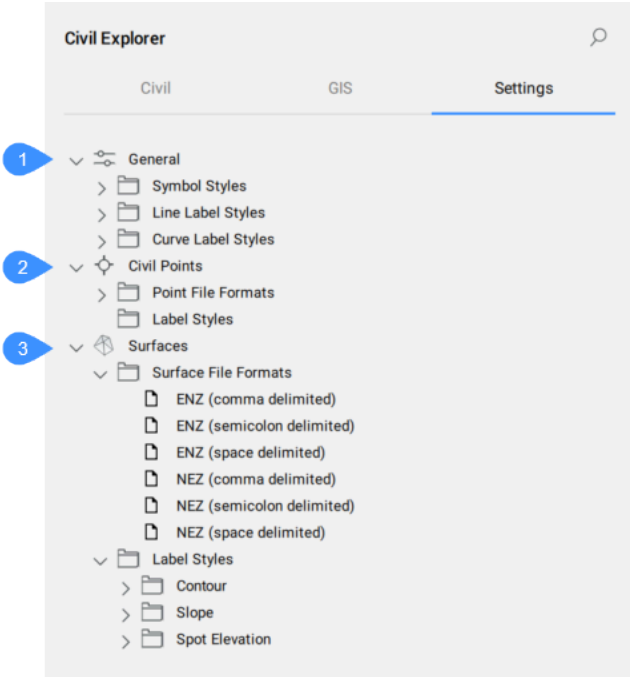
Rename group
Renames the group.

Add sub-group
Adds a sub-group.

Remove group
Removes the group.

8.24.4 Settings Tab

The **Settings** tab manages the civil entity styles and formats.



1 General

2 Civil Points

3 Surfaces

General
Defines the general symbol and label styles for civil entities.

Symbol styles

Lists predefined and custom-created symbol styles. You can **Create**, **Edit**, **Copy**, or **Delete** a symbol style. Click the corresponding option from the context menu to open the **Symbol Style** dialog box.

Line Label Styles

Lists predefined and custom-created line label styles.

Curve label Styles

Lists predefined and custom-created curve label styles.

The context menu contains the **Create**, **Edit**, **Copy**, or **Delete** options. Click the corresponding option to open the **Label Style Editor** dialog box.

Civil Points

Defines civil points file formats and label styles for civil points.

Point File Formats

Lists predefined and custom-created point file formats. The list can be managed by double-clicking or pressing the **Edit** option from the context menu which opens the **Manage Point File Formats** dialog box.

Predefined point file formats:

- NameENZ (comma delimited)
- NameENZ (semicolon delimited)
- NameENZ (space delimited)
- NameENZD (comma delimited)
- NameENZD (semicolon delimited)
- NameENZD (space delimited)
- NameNEZ (comma delimited)
- NameNEZ (semicolon delimited)
- NameNEZ (space delimited)
- NameNEZD (comma delimited)
- NameNEZD (semicolon delimited)
- NameNEZD (space delimited)
- PENZ (comma delimited)
- PENZ (semicolon delimited)
- PENZ (space delimited)
- PENZD (comma delimited)
- PENZD (semicolon delimited)
- PENZD (space delimited)
- PNEZ (comma delimited)
- PNEZ (semicolon delimited)
- PNEZ (space delimited)
- PNEZD (comma delimited)
- PNEZD (semicolon delimited)

- PNEZD (space delimited)

where:

- P is Point Number
- E is Easting
- N is Northing
- Z is Elevation
- D is Raw Description
- Name is for points that contain alphanumeric values for the point number. Also, the point name supports alphanumeric values.

Label Styles
Lists the available label styles for Civil Points. The context menu allows you to **Create**, **Edit**, **Copy**, or **Delete** a label style. **Create**, **Edit** and **Copy** options open the **Label Style Editor** dialog box which allows you to define a new label style.

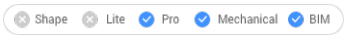
Surfaces
Defines surface file formats and label styles for Contour, Slope and Spot elevation.

Surface File Formats
Lists predefined and custom-created surface file formats. The list can be managed by double-clicking or pressing the **Edit** option from the context menu that opens the **Manage Point File Formats** dialog box.

Label styles
Lists predefined and custom-created styles for Contour, Slope and Spot elevation. The context menu allows you to **Edit**, **Copy**, or **Delete** a label style. The **Edit** and **Copy** options open the **Label Style Editor** dialog box which allows you to define a new label style.

8.25 -CIVILPOINT command

Creates Civil Points via Command line.



8.25.1 Description

Civil Points are defined in BricsCAD as Civil entities, which are represented by symbols and labels. The graphical representation of Civil points can be defined through **Symbol style** and **Label style**. In addition to XYZ coordinates, Civil Points can have a variety of attributes assigned, including point number, point name, raw (field) description and full (expanded) description. You can also add any other attributes to Civil points and display them in the point label style.

Use the **Properties** panel to edit the properties of an individual point or a group of selected points.

For improved organization, Civil Points can be collected into point groups based on certain criteria determined by filters.

You can use the basic BricsCAD commands to edit Civil Points in a drawing. For example, COPY, COPYCLIP, PASTE, PASTECLIP, MOVE, ROTATE.

You can use Civil Points as input objects for creating a TIN Surface, which is automatically updated when editing points.

8.25.2 Method

There are two methods to insert Civil Points:

- Insert a single point by specifying its location in a drawing.
- Import multiple points from an ASCII point file.
- ***Note:*** In addition to the basic ASCII file, points can also be imported from a LandXML file using the LANDXMLIMPORT command.

8.25.3 Options within the command

Specify point location
Allows you to specify the location of a new individual point in a drawing.

Enter point description
Allows you to specify a description for the Civil Point.

Enter point elevation
Allows you to specify an elevation for the Civil Point.

Import points from file
Creates Civil Points from an imported points file in text file format (TXT), comma-delimited file format (CSV), and other text formats where the XYZ coordinates in the input file are separated by any delimiter.

Enter file name
Allows you to specify the file path name.

Enter file format name
Allows you to specify the point file format.

set Style
Allows you to specify a point symbol style when inserting a new point.

Note: More information on point styles and labels can be found in the **Working with Civil points** article.

?
Lists the names of the available point symbol styles in the Command line.

8.26 CIVILPOINT command

Creates Civil Points.

Shape Lite Pro Mechanical BIM

Icon: 

8.26.1 Description

Civil Points are defined in BricsCAD as Civil entities, which are represented by symbols and labels. The graphical representation of Civil points can be defined through **Symbol style** and **Label style**. In addition to XYZ coordinates, Civil Points can have a variety of attributes assigned, including point number, point name, raw (field) description and full (expanded) description. You can also add any other attributes to Civil points and display them in the point label style.

Use the **Properties** panel to edit the properties of an individual point or a group of selected points.

For improved organization, Civil Points can be collected into point groups based on certain criteria determined by filters.

You can use the basic BricsCAD commands to edit Civil Points in a drawing. For example, COPY, COPYCLIP, PASTE, PASTECLIP, MOVE, ROTATE.

You can use Civil Points as input objects for creating a TIN Surface, which is automatically updated when editing points.

8.26.2 Method

There are two methods to insert Civil Points:

- Insert a single point by specifying its location in a drawing.
- Import multiple points from an ASCII point file.
- ***Note:*** In addition to the basic ASCII file, points can also be imported from a LandXML file using the LANDXMLIMPORT command or converted from Civil 3D drawing using the CIVIL3DIMPORT command.

8.26.3 Options within the command

Specify point location
Allows you to specify the location of a new individual point in a drawing.

Enter point description
Allows you to specify a description for the Civil Point.

Enter point elevation
Allows you to specify an elevation for the Civil Point.

Import points from file
Imports Civil Points from a point file in text file format (TXT), comma-delimited file format (CSV) and any other ASCII format where the point attributes in the columns are separated by any delimiter. You can select multiple point files from the **Import Points From Point File** dialog box which allows you to import Civil Points from an ASCII point file and to select the appropriate point file format.

Note: The UTF-8, UTF-8-BOM and ANSI encoding are supported when creating Civil Points from point files.

set Style
Allows you to specify a point symbol style when inserting a new point.

Note: More information on point styles and labels can be found in the **Working with Civil points** article.

?
Lists the names of the available point symbol styles in the Command line.

8.27 CIVILPOINTATTRIBUTES command

Adds or removes user-defined attributes to Civil Points.

Shape Lite Pro Mechanical BIM

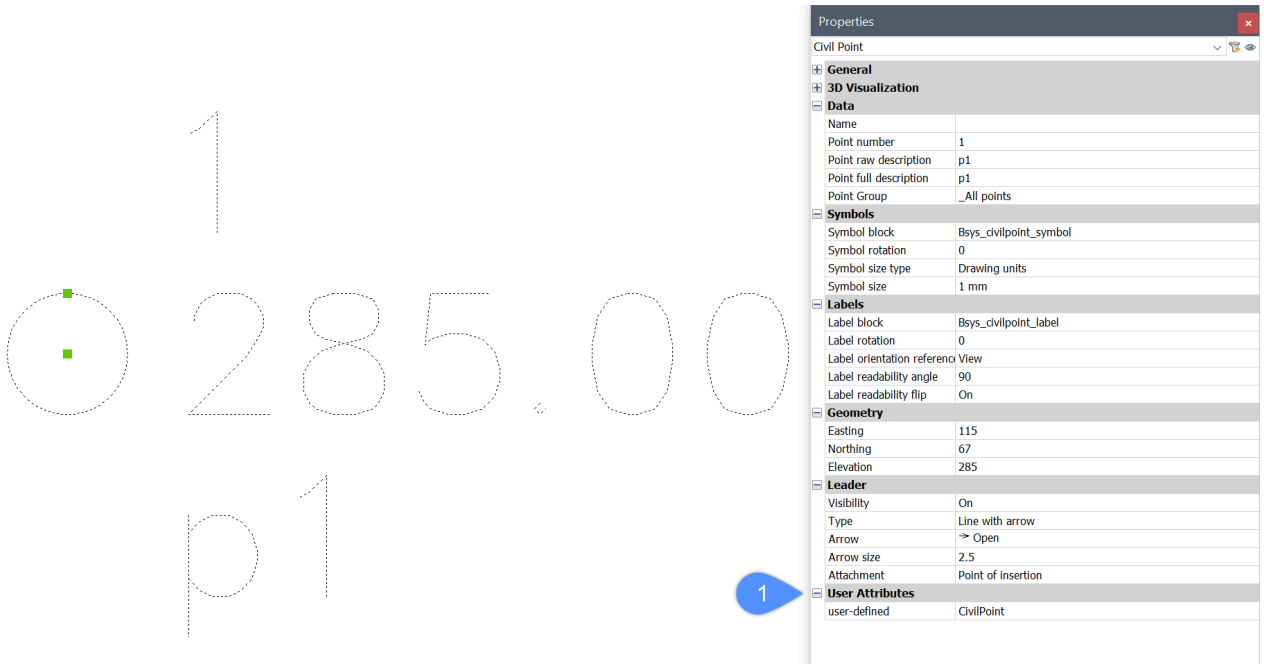
Icon:

8.27.1 Options within the command

Select civil points
Allows you to select Civil Points to add user-defined attributes.

? to list user attributes
Lists the existing user-defined attributes on Civil Points in the Command line.

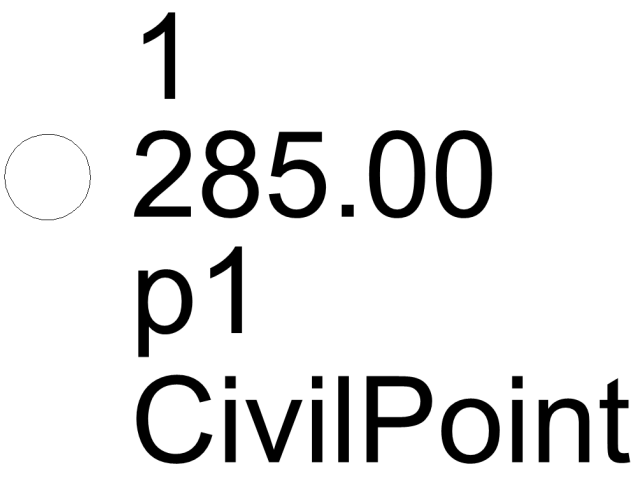
Add attribute
Adds attributes for the selected Civil Point by specifying a key (name) and a value.
User defined attributes are added to the **User Attributes** (1) section in the **Properties** panel.



You can use the block editor to add a new custom attribute to the Civil point **Symbol block**. The new attribute is added by writing the attribute key in angle brackets: **<user-defined>**. The attribute key can be written as Text or as an Attribute definition.
User defined attribute added in the block editor:

<Number>
<Elevation>
<Full Description>
<user-defined>

The Civil point Symbol block displayed in the Layout view:



Remove attribute
Removes user-defined attributes on Civil Points based on the specified attribute keys.

8.28 CIVILPOINTEDIT command

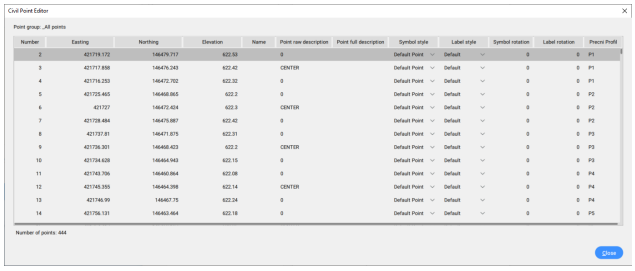
Edits Civil points.

Shape Lite Pro Mechanical BIM

8.28.1 Description

Opens the **Civil Point Editor** dialog box that allows you to edit the selected Civil points.

The **Civil Point Editor** dialog box displays Civil points and their properties in a table.



Note:

- Click a cell to edit the content.
- It is possible to sort the Civil points by some of their properties.
- The **Civil Point Editor** dialog box is modeless. This means you can work in the drawing while the **Civil Point Editor** dialog box is available on the screen.

Right-click the selected point(s) to open a context menu:

Zoom to
Zooms in to the selected civil point.

Pan to
Pans to the selected civil point in the drawing.

Select
Selects the civil point(s).

Copy to clipboard
Copies the civil point(s) to the clipboard.

Delete point
Removes the civil point(s).

Note: A warning dialog box opens to specify whether you are sure to delete the points.

8.29 -CIVILPOINTEXPORT command

Exports Civil points to an external file via Command line.

Shape Lite Pro Mechanical BIM

8.29.1 Method

Select Civil Points in the drawing, indicate the point file format name, then the file name and location.

Note: The **Export As** dialog box opens when the FILEDIA system variable is ON.

8.29.2 Options within the command

point Group
Specifies the point group name to export.

? to list point groups
Lists all available point groups in the drawing. You can copy/paste the name of it in the Command line.

Enter point file format name
Specifies the point file format name used for export.

? to list point file formats
Lists all available point file format names. You can copy/paste the name of it in the Command line.

Include additional point attributes that are not defined in the selected format
Allows you to choose whether to include point attributes that are not defined in the selected format.

8.30 CIVILPOINTEXPORT command

Exports Civil points to an external file.

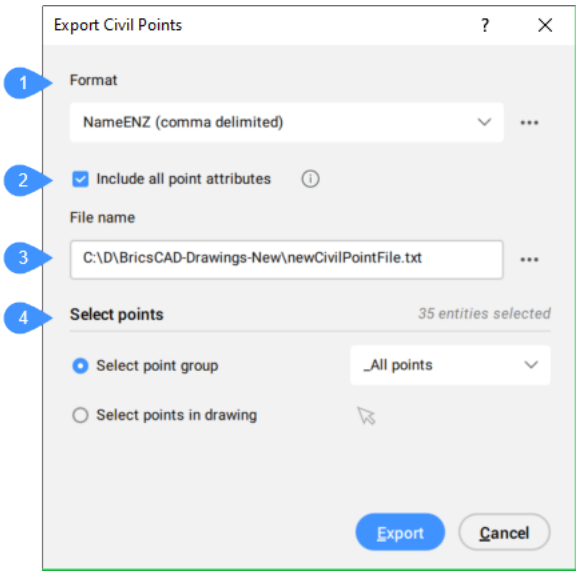
Shape Lite Pro Mechanical BIM

Icon: 

8.30.1 Description

Opens the **Export Civil Point** dialog box.

The **Export Civil Points** dialog box allows you to export the Civil Points to an external file.



1 Format

2 Include all point attributes

3 File name

4 Select points

Format
Allows you to select from the drop-down list a format for the exported file.

Note: Press the three points from the right side to open the **Manage Point File Formats** dialog box to create a new file format.

Include all point attributes
If checked, includes additional point attributes that are not defined in the selected format for export.

File name
Defines the path and file name.

Select points
Allows you to select Civil Points for export.

Select point group
Allows you to select a point group from the drop-down list.

Select points in drawing

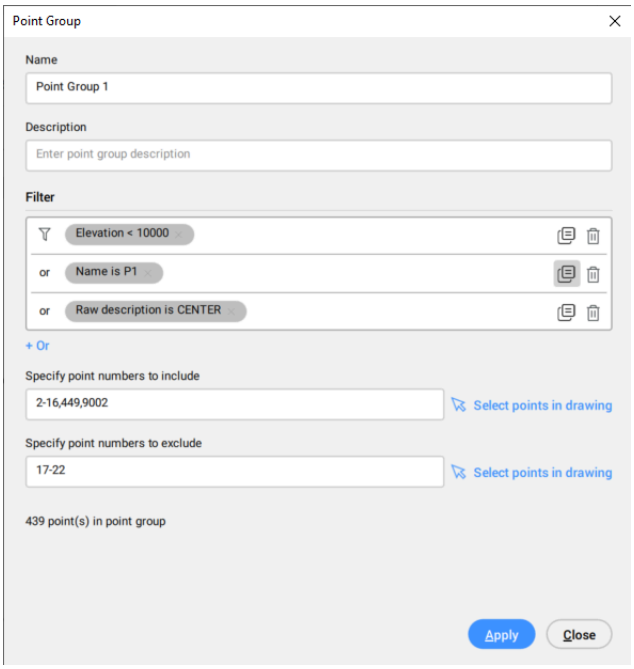
Click the select button ( ) to select Civil Points in the drawing for export.

8.31 CIVILPOINTGROUP command

Creates a civil point group.

8.31.1 Method

Opens the **Point Group** dialog box that allows you to create or edit a Civil Point Group.



Note: Click a cell to edit the content.

Name

Allows you to enter the name of a new point group for which you will create a new filter or display the name of an existing point group.

Note: If the point group name is not defined, a warning message appears.

Description

Allows you to enter a description of the point group or display the description of an existing point group.

Filter

Allows you to create a new or to edit an existing filter in which you specify the criteria for including and/or excluding points in the selected point group.

Note: You can add, duplicate or delete the filters.

Note: Once an individual filter is created in a particular drawing, it also affects all the points that are inserted later in that drawing.

The parameters used for filtering are:

- Elevation
- Name
- Raw description
- Full description
- User attribute key
- User attribute value

Specify point numbers to include

Allows you to enter the point numbers to be included in the specified filter.

Note: Points can be selected from the drawing by pressing the arrow from the right side of the field.

Specify point numbers to exclude
Allows you to enter the point numbers to be excluded from the specified filter.

Note: Points can be selected from the drawing by pressing the arrow from the right side of the field.

Note:

- If no filter is defined, a warning message appears.
- It is possible to create a point group even if no Civil points are present in the drawing.

8.32 -CIVILPOINTGROUP command

Creates **Civil Point** groups via Command line.

Shape Lite Pro Mechanical BIM

Icon: 

8.32.1 Description

Depending on the specified filters, points can be included or excluded from the point group.

8.32.2 Method

There are two ways to use the command:

- Create a new point group and add filters to include or exclude points from the group.
- Edit filters of an existing point group.
- With this method, you can specify the name of the existing point group in the first step and then you can edit the filters.

8.32.3 Options within the command

Enter point group name
Allows you to enter the name of a new point group for which you will create a new filter. It also allows you to enter the name of an existing point group for which you want to edit the filters.

filter to Include
Allows you to create a new or to edit an existing filter in which you specify the criteria for including points in the selected point group.

1 Select the name of existing Civil Point attribute in the Command line.

2 Specify either an individual attribute value, multiple values, a range of values, or any combination of the listed options, based on which points are included in the desired point group. When specifying multiple attribute values, a comma (,) without a space is used for separation.

In addition to entering full attribute values/keys, you can define filters in other ways:

- Only specify the first character of the name and the "*" sign, such as "a*", where the filter will take into consideration all names starting with "a".
- Specify the first and last character with an intermediate sign "*", such as "a*z", where the filter will take into consideration all names starting with "a" and ending with "z".

- Specify sign "*" and a last character, such as "*z", where the filter will take into consideration all names ending with "z".
- Specify individual characters, missing characters are replaced with the "*" sign, such as "a*b*" where the filter will take into consideration all names starting with "a", and with the third character "b".

Note:

- Multiple filters can be added to the existing point group.
- Once an individual filter is created in a particular drawing, it also affects all the points that are inserted later in that drawing.

filter to Exclude
Allows you to create a new or to edit an existing filter in which you specify the criteria for excluding points from the selected point group.
The procedure of adding a new filter is exactly the same as described in the **filter to Include** section.

? to list point groups
Lists existing point groups in the drawing on the Command line.

Specify point numbers to include
Allows you to enter the point numbers to be included in the specified filter.

Specify point numbers to exclude
Allows you to enter the point numbers to be excluded from the specified filter.

Name
Allows you to enter the point names to be included in the specified filter, if the option **filter to Include** is selected in the previous step.
Allows you to enter the point names to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

Note: **_All points** cannot be excluded or edited.

Elevation
Allows you to enter the point elevations to be included in the specified filter, if the option **filter to Include** is selected in the previous step.
Allows you to enter the point names to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.
An example of an Elevation filter with multiple attribute values, including a range of values, looks like this: "100-200,>400". This filter includes all points with an elevation range between 100 and 200 meters and at the same time all points with an elevation higher than 400 m.
Let's add a new filter to our sample point group to include points based on points raw description: "Manhole".
Based on the additional filter, only points that meet criteria of both filters: Elevation="100-200,>400" and Raw Description="Manhole" are included in the example point group.

Full description
Allows you to enter the point full descriptions to be included in the specified filter, if the option **filter to Include** is selected in the previous step.
Allows you to enter the point full descriptions to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

Raw description

Allows you to enter the point raw descriptions to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the point raw descriptions to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

user attribute Key

Allows you to enter the user-defined attribute keys (attribute names) to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the user-defined attribute keys (attribute names) to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

user attribute Value

Allows you to enter the user-defined attribute keys (attribute names) to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the user-defined attribute keys (attribute names) to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

Remove group

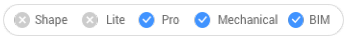
Removes the selected civil point group.

reName group

Allows you to rename the selected civil point group.

8.33 CIVILPOINTZOOM command

Provides a quick way to navigate to a civil point.



8.33.1 Description

Zooms in to a civil point by indicating its number or name.

8.33.2 Options within the command

point Name

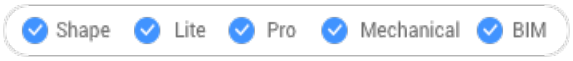
Defines a civil point by its name.

point Number

Defines a civil point by its point number.

8.34 CLEANSCREENOFF command

Displays user interface elements that were hidden by the CLEANSCREENON command.



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

8.35 CLEANSCREENON command

Makes the drawing area larger by hiding elements of the user interface.

Shape Lite Pro Mechanical BIM

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

8.35.1 Method

Elements in the user interface are hidden according to the values of the CLEANSCREENOPTIONS system variable.

8.36 CLEANUNUSEDVARIABLES command

Purges parametric variables not used by constraint expressions and not linked to dimensions.

Shape Lite Pro Mechanical BIM

8.36.1 Description

Automatically deletes parametric variables not used by constraint expressions and not linked to dimensions when the command is run.

8.37 CLIP command

Clips the display of the selected entity to a specified clipping boundary.

Shape Lite Pro Mechanical BIM

8.37.1 Method

Select an entity to clip it by defining a clipping boundary.

The list of options in the Command line varies depending on the type of the selected entity. You can select one of the following entities: Xref, block, image, PDF, or viewport.

Note: The CLIP command unifies the XCLIP, IMAGECLIP, PDFCLIP and VPCLIP commands.

Clipping an Xref or block reference

Options within the Command line when clipping an Xref or block reference are described in the **XCLIP command** article.

Clipping an image

Options within the Command line when clipping an image are described in the **IMAGECLIP command** article.

Clipping a PDF

Options within the Command line when clipping a PDF are described in the **PDFCLIP command** article.

Clipping a viewport

Options within the Command line when clipping a viewport are described in the **VPCLIP command** article.

8.38 CLIPDISPLAY command

Toggles the Clip Display property of section planes and BIM section entities.

Shape Lite Pro Mechanical BIM

Icon:

Note: The display of the section entity and be toggled also through the **Properties** panel.

8.39 CLIPIT command (Express Tools)

Clips images, wipeouts, blocks or external references.

Shape Lite Pro Mechanical BIM

Icon:

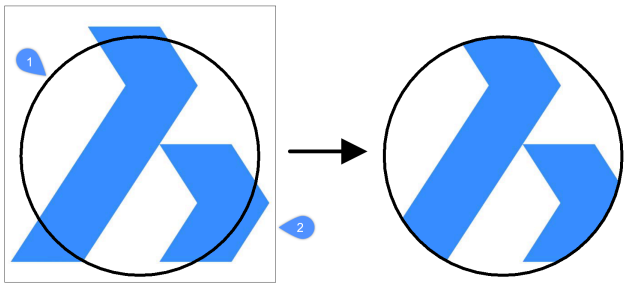
8.39.1 Description

Clips images, wipeouts, blocks or external references by a clipping edge.

Note: Only polyline, circle, arc, ellipse or text entities can be used as a clipping edge.

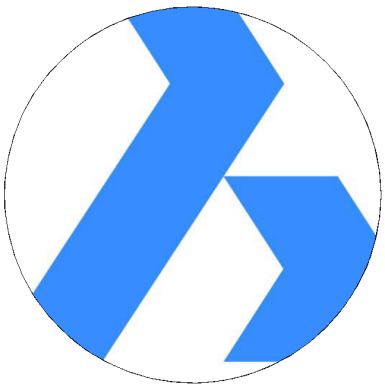
8.39.2 Method

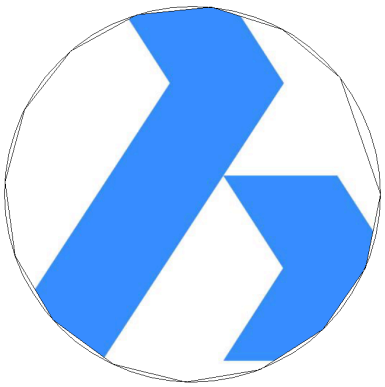
Select the clipping edge (1), then the entity to clip (2).



Enter the maximum allowable error distance for the resolution of arc segments.

Note: The default value is 0.02. A lower value gives a smoother clipping but results in a slower performance of BricsCAD when a drawing is regenerated.

- Error distance = 0.02
- 
- Error distance = 1



8.40 CLOSE command

Closes the current drawing.

Shape Lite Pro Mechanical BIM

8.40.1 Description

Closes the current drawing after it has been saved. If changes have been made since the last save, a **BricsCAD** dialog box offers you the opportunity to save the drawing before closing it.

8.41 COLOR command

Opens the **Color** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

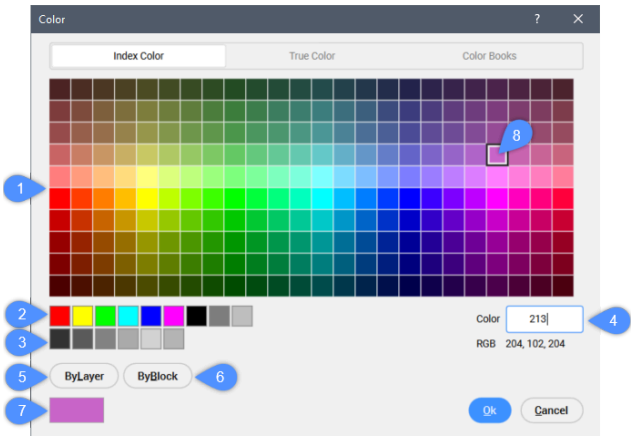
Alias: COL, COLOUR, DDCOLOR, DDCOLOUR, SETCOLOR

8.41.1 Description

Opens the **Color** dialog box to specify the current color.

The **Color** dialog box allows you to select a color for layers, grids, dimensions, background, etc. It contains 3 tabs: **Index Color**, **True Color** and **Color Books**.

8.41.2 Index Color tab



Color palette (1)
Displays 240 colors.

Basic colors (2)
Displays the basic colors.

Grey colors (3)
Displays shades of gray.

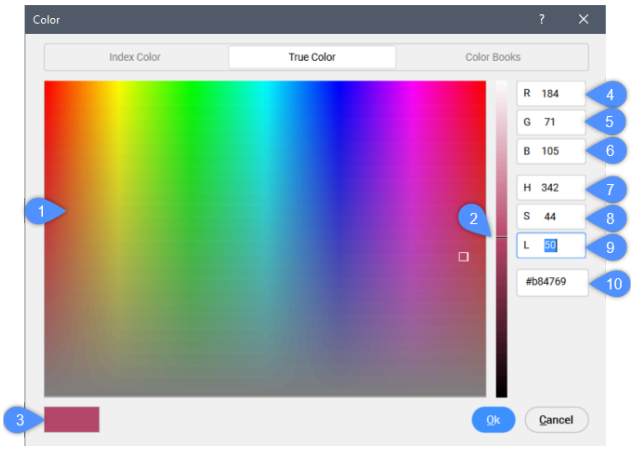
Color index (4)
Displays the index of the selected color. You can insert an index and preview the color in field (7). The corresponding color is marked with a square (8).

ByLayer (5)
Sets the color to ByLayer in which entities adopt the color defined by their layer's property.

ByBlock (6)
Sets the color to ByBlock in which entities adopt the color defined by their block.

Color sample (7)
Previews the selected color.

8.41.3 True Color tab



Color (1)
Specifies hue (left and right) and saturation (up and down).

Luminosity bar (2)
Specifies the luminosity of the color.

Color sample (3)
Previews the selected color.

Red (R) - (4)
Specifies the amount of red in the color. It ranges from 0 (no red) to 255 (full red).

Green (G) - (5)
Specifies the amount of green in the color. It ranges from 0 (no green) to 255 (full green).

Blue (B) - (6)
Specifies the amount of blue in the color. It ranges from 0 (no blue) to 255 (full blue).

Note: Hue, saturation and luminosity work together, while red, green and blue work separately.

Hue (H) - (7)
Specifies the color's hue. It ranges from red to green, blue and pink.
It ranges from 0 to 359.

Saturation (S) - (8)
Specifies the intensity of the color. It ranges from 0 to 100. Saturation is the intensity of the color, where 0 = gray and 100 = full color.

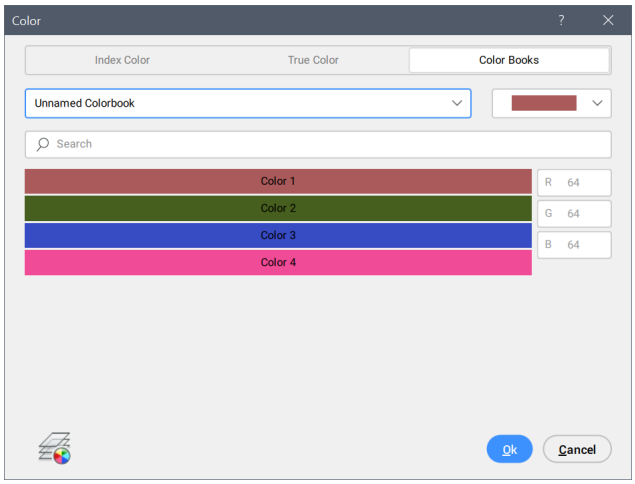
Luminosity (L) - (9)
Specifies the brightness of the color. It ranges from 0 to 100. Luminosity is the lightness or darkness of the color, where 0 = black and 100 = white.

Note: When the current color is white (L=100) or black (L=0) and a true color is picked on the **COLOR** dialog box, the luminosity is set to 50. Also, the picked true color is not synced with the luminosity.

Color code (10)
Displays the selected color code.

8.41.4 Color Books tab

Choose a color book that is present on your system.



Note:

- The COLORBOOKPATH system variable specifies the folder in which BricsCAD should look for color book files.
- Encrypted color books are supported. The encrypted color books are not included within BricsCAD.

8.42 -COLOR command

Sets the current working color, through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -COL, -COLOUR

8.42.1 Methods

Enter the name, number, or RGB value of a color:

- Specify a color name: Red, Yellow, Green, Cyan, Blue, Magenta, White, ByLayer, or ByBlock.
- Specify a number from 0 to 256, which represents the Index Color.
- Specify a value for each of red, green, and blue, which represents the True Color. The range is from 0 to 255. For example, white is 255,255,255 and gray is 128,128,128.
- Specify a color book name.

8.43 COMMANDLINE command

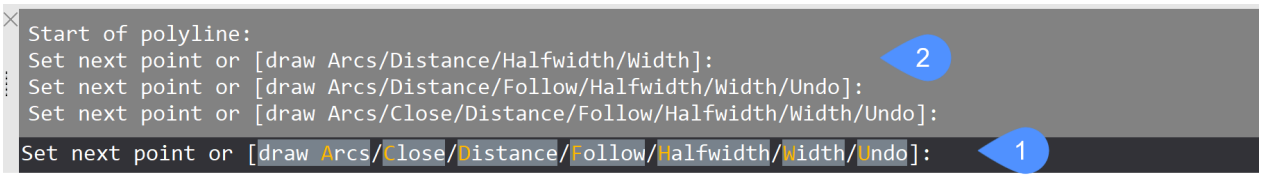
Opens the **Command line** panel.

Shape Lite Pro Mechanical BIM

8.43.1 Description

Opens the **Command Line** panel to display it in the current workspace. The **Command line** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Command Line** can be either floating, docked or stacked.

The **Command line** panel allows you to launch commands or modify system variables by typing the command or variable name.



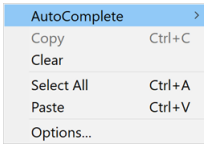
1 Command line

2 Command history

8.43.2 Command line

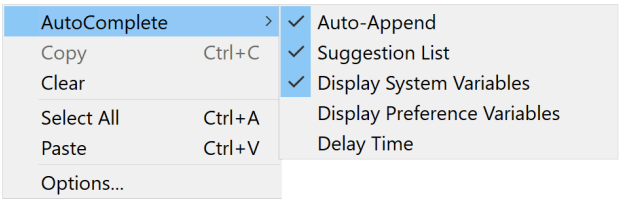
Displays the current command prompt. If no command is active, it prompts you to Enter Command. If a command is active, it displays prompts and options relevant to the current command. Prompt shortcuts are highlighted. You may enter the shortcut characters in the Command line or click on them with the 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.

8.43.3 Command History

Displays the history of the Command line input for the current BricsCAD session.

8.43.4 Accessing the input history

Use the Up Arrow and Down Arrow keys at the Command prompt or at an input prompt to access and reuse the history of recent commands input.

Note: Markers for recent input of point locations are displayed in the drawing area. The markers are controlled by SNAPMARKERSIZE, SNAPMARKERCOLOR, and SNAPMARKERTHICKNESS system variables.

Remember: The input history can include the last 12 items from the current command (from the current and previous calls) and the last 12 items from other previous commands.

8.44 COMMANDLINEHIDE command

Closes the **Command Line** panel.

Shape Lite Pro Mechanical BIM

8.44.1 Description

Closes the **Command Line** panel to hide it from the current workspace. If the **Command Line** panel is stacked when you close it, the Command line tab or icon is removed from the stack.

8.45 COMMANDS command

Lists the names of the commands.

Shape Lite Pro Mechanical BIM

8.45.1 Description

Lists the names of commands available in the program according to your license level, first the name in English followed by the localized command name.

8.45.2 Options

Internal
Lists command names native to BricsCAD.

External
Lists command names from add-on applications.

Both
List both internal and external command names.

8.46 COMMUNICATORINFO command

Checks whether the Communicator for BricsCAD® add-on is correctly installed and provides a diagnostic report.

Shape Lite Pro Mechanical BIM

Note: The command is available on Windows only.

Until V21, the major versions of BricsCAD and Communicator for BricsCAD® should match (for example: Communicator for BricsCAD® V20.2.x works with BricsCAD V20.2.x).

Starting with V21 the compatibility between minor releases is supported (for example:Communicator for BricsCAD® V21.1.x will work with BricsCAD V21.2.x).

8.46.1 Description

Switches to the Prompt History window and then provides a report.

When Communicator for BricsCAD® is not installed, then the command reports:

```
Brief check result: Communicator module loading: FAILED
```

8.47 CONE command

Creates a 3D Solid in the shape of a cone.

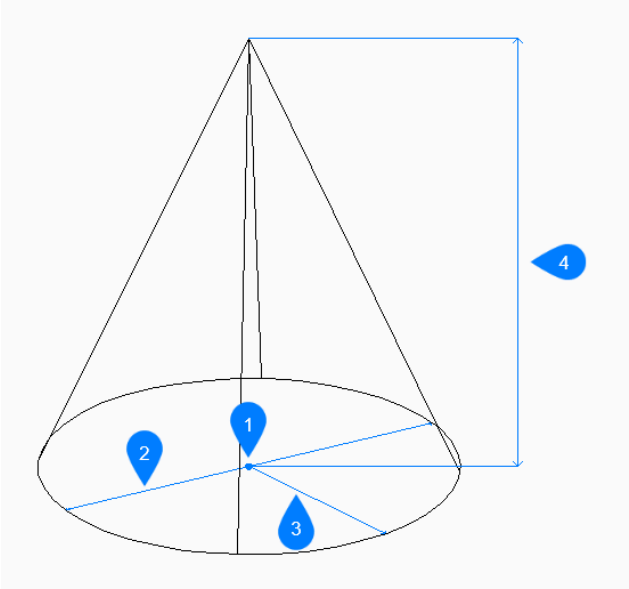
Note: In BricsCAD Lite, which doesn't support 3D solids, the CONE command launches the AI_CONE command.

Shape Lite Pro Mechanical BIM

Icon: 

8.47.1 Description

Creates a 3D solid in the shape of a circular or elliptical cone. Choose from a combination of options including center, radius, diameter, 3 point, 2 point, tangents, axis endpoints, and height.



1 Center

2 Diameter

3 Radius

4 Height

8.47.2 Method

This command has 5 methods to begin creating a cone:

- Center point
- 3 Point
- 2 Point
- Tangent tangent radius
- Elliptical

8.47.3 Options within the command

Select center point
Allows you to begin creating a circular cone by specifying the center of the base.

Radius of base of cone
Allows you to specify the radius of the base of the cone.

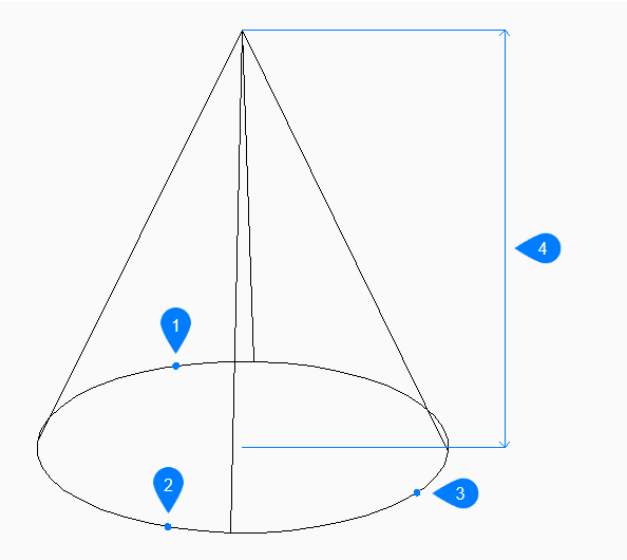
Diameter
Allows you to specify the diameter of the cone's base.

3Point
Allows you to begin creating a circular cone by specifying three points on the circumference of its base.

First point
Specifies the first point on the circumference.

Second point
Specifies the second point on the circumference.

Third point
Specifies the third point on the circumference.



1 Point 1

2 Point 2

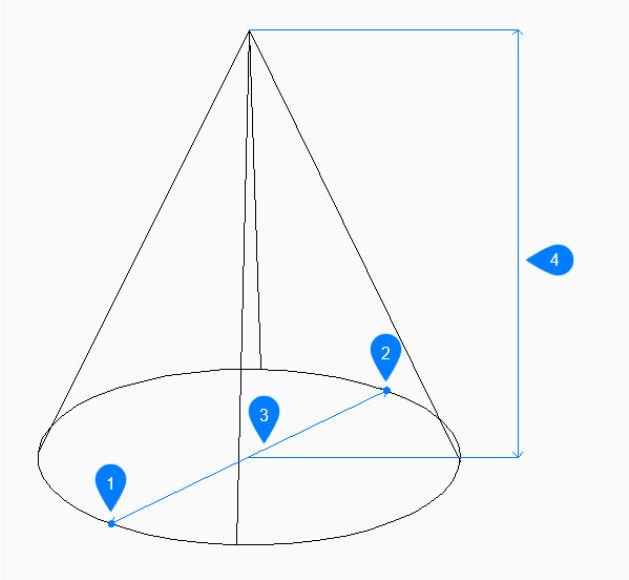
3 Point 3

4 Height

2Point
Allows you to begin creating a circular cone by specifying two points on the circumference of its base.

First end of diameter
Specifies the first end of diameter.

Second end of diameter
Specifies the second end of diameter.



1 Point 1

2 Point 2

3 Diameter

4 Height

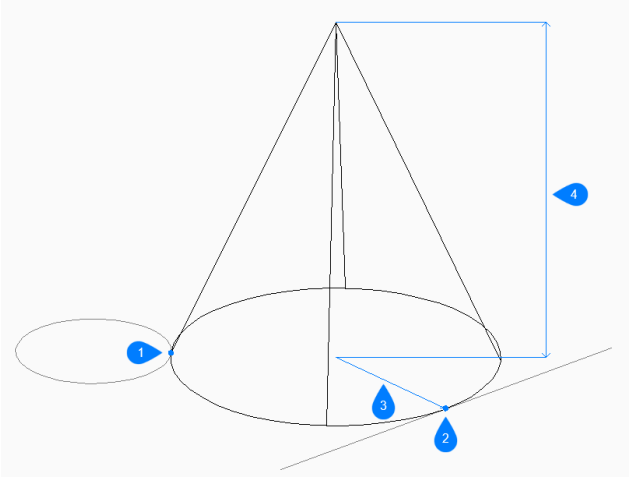
Tangent Tangent Radius
Allows you to begin creating a circular cone by selecting a tangent point on the first and second entity.

Specify point on object for first tangent
Allows you to select a tangent point on the first entity.

Specify point on object for second tangent
Allows you to select a tangent point on the second entity.

Radius of circle
Allows you to specify a valid radius for the base.

Note: If you specify a radius that is not possible with the selected tangents, you are prompted to specify the tangents and radius again.



1 Tangent point 1

2 Tangent point 2

3 Radius

4 Height

Elliptical

Allows you to begin creating an elliptical cone by specifying the first end, second end and the other end of the ellipse axis.

Set first end of ellipse

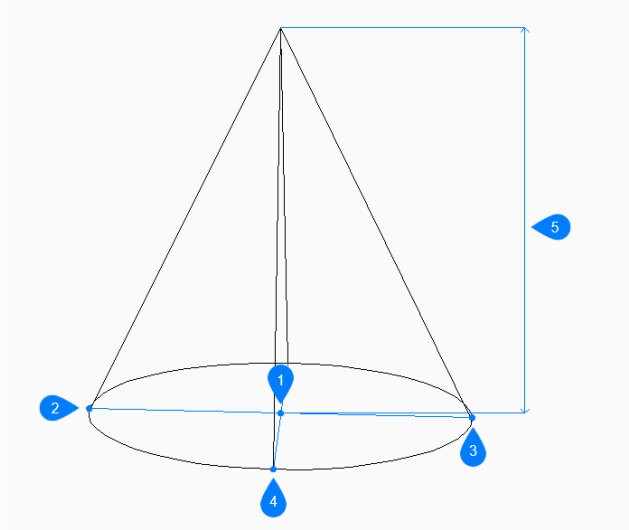
Specifies the first end of the ellipse axis.

Second end of ellipse axis

Specifies the second end of the ellipse axis.

Other end of ellipse axis

Specifies the radius of the other ellipse axis.



1 Center

2 First end of ellipse axis

3 Second end of ellipse axis

4 End of other axis

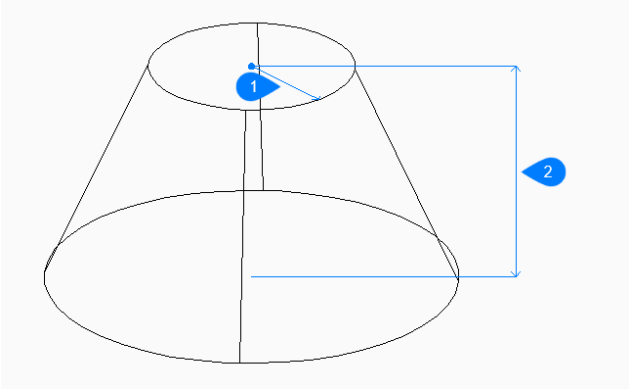
5 Height

Specify height
Allows you to specify the height of the cone.

2Point
Specifies the height of the cone as the distance between any two points.

Axis endpoint
Specify the axis endpoint to define the height and orientation of the cone in 3D space. The center of the base is used as the other axis endpoint.

Top radius
Specify a radius for the top of the cone. Any radius larger than zero (0) creates a cone with a flat top.



1 Top radius

2 Height

8.48 CONNECT command

Connects coplanar lines, arcs and/or polylines.

Shape Lite Pro Mechanical BIM

Icon:

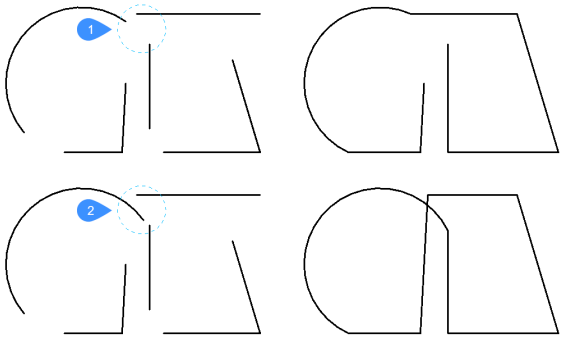
8.48.1 Description

Allows you to connect one or more coplanar lines, arcs and/or polylines whose start and/or end points don't overlap by trimming and extending as needed. The connected entities are then joined into one or more polylines, when possible.

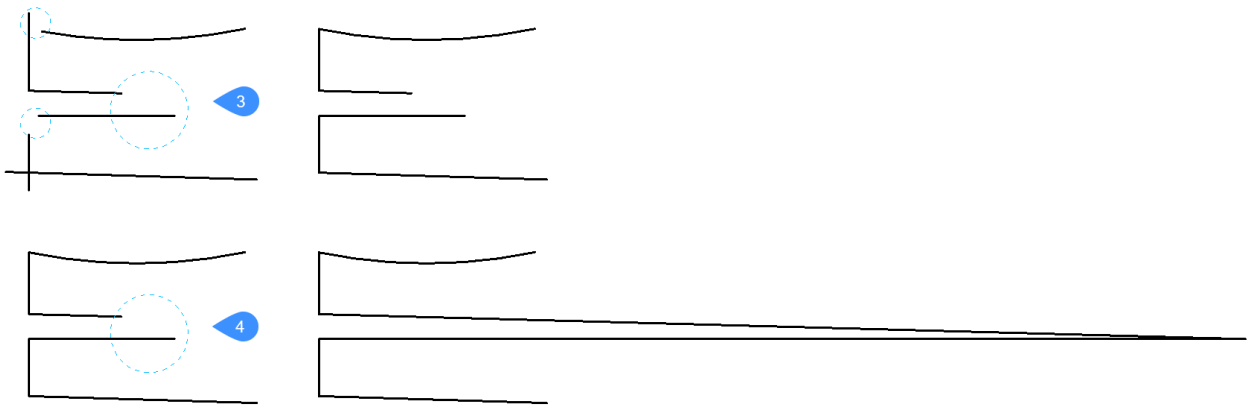
8.48.2 Method

The command finds and hierarchically closes the gaps between the selected entities. Based on the distribution of gap sizes, small gaps are closed first (see differences between (1) and (2) and the corresponding results). Entities whose extensions do not intersect are ignored.

Note: The underlying geometric definition of the input entities is not changed.

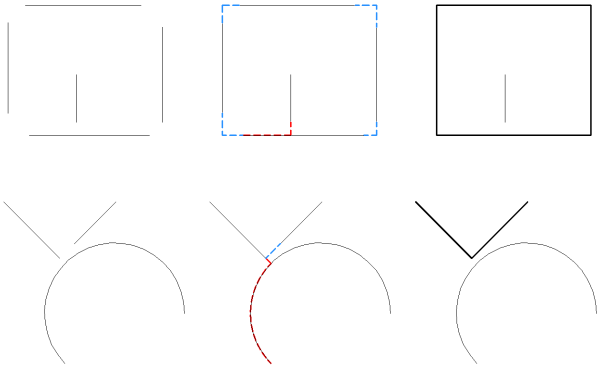


Running the command a second time on the same selection set can result in additional gaps being closed (e.g. gaps between nearly parallel entities (4)) that were previously considered much larger than the other gaps (3).



The CONNECT command can also be applied to collinear lines, to combine these into a single line or polyline segment. There can be exceptions when multiple (pairs of) lines converge in the same point. In all cases, smaller gaps are closed first.

Note: The gap size between two entities (lines and/or arcs) is defined as the sum of the distances between the current end points of the entities and the possible connection point. This is exemplified in the images below that show small gaps (in blue) and big gaps (in red), and the command's results for the two situations.



8.48.3 Options within the command

Select entities
Lets you select entities to connect.

Entire drawing
Selects all entities in the drawing to connect.

selection options (?)
Lists all additional selection methods.

8.49 CONSTRAINTBAR command

Shows and hides constraint bars.

Shape Lite Pro Mechanical BIM

Icon: 

8.49.1 Description

Shows and hides constraint bars next to entities constrained with geometric constraints.

Note: Constraint Bars are initially hidden when a drawing is opened.

8.49.2 Options

Show
Displays the constraint bar next to the entities you selected.

Hide
Hides the constraint bar next to the entities you selected.

Reset
Relocates constraint bars to their default positions, which is near the midpoint of the entity.

8.50 CONTENTBROWSERCLOSE command

Closes the **Content Browser** panel.

Shape Lite Pro Mechanical BIM

8.50.1 Description

Closes the Content browser panel to hide it from the current workspace. If the **Content Browser** panel is stacked when you close it, the Content browser tab or icon is removed from the stack.

8.51 CONTENTBROWSEROPEN command

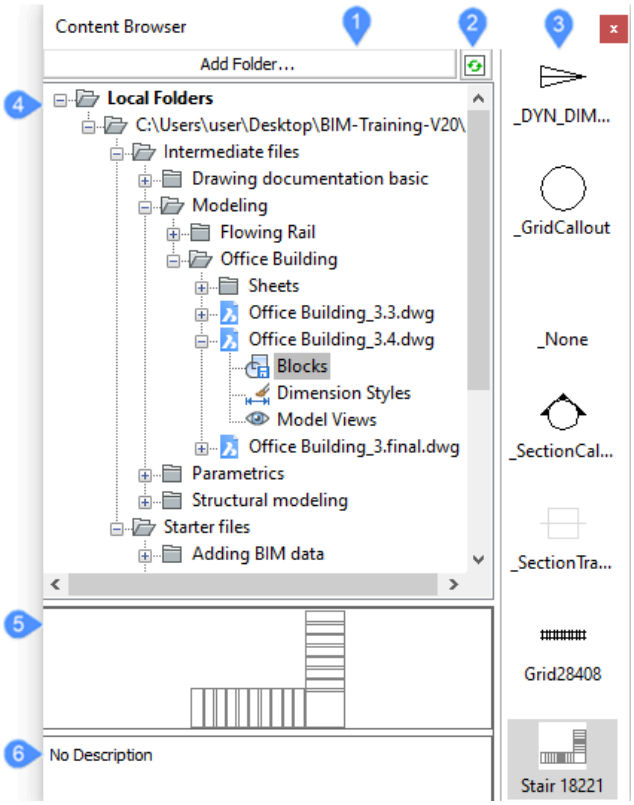
Opens the **Content Browser** panel.

Shape Lite Pro Mechanical BIM

8.51.1 Description

Opens the **Content Browser** panel to display it in the current workspace. The **Content Browser** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Content Browser** panel can be either floating, docked or stacked.

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

8.51.2 Add folder...

Displays the **Choose a folder** dialog box.

8.51.3 Refresh

Select a drawing and press the Refresh button to expand its content (Blocks, Dimension Styles, Model Views).

8.51.4 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.

You can drag & drop a dimension style into the current drawing. If a dimension style with the same name already exists, a **Copy / Paste** dialog box will appear.

You can drag & drop a model view into the current drawing. This option is available only in Paper Space.

8.51.5 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.

8.51.6 Definitions' preview

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

8.51.7 Description

Displays a description of your selected definition.

8.52 CONVERTCOMPONENTSTOBLOCKS command

Automatically converts legacy and mechanical components in BricsCAD® to mechanical blocks.

Shape Lite Pro Mechanical BIM

8.52.1 Description

Automatically converts legacy and mechanical components in BricsCAD® to mechanical blocks while preserving all their parameters and constraints. Also, it converts arrays of legacy and mechanical components into mechanical blocks.

A report about the conversion displays in the Command line.

8.53 CONVERTCTB command

Opens the **Select color-dependent plot style table** file dialog box.

Shape Lite Pro Mechanical BIM

8.53.1 Description

Opens the **Select color-dependent plot style table file** dialog box to select a CTB file to convert to an STB file.

8.54 CONVERTPOLY command

Converts 2D and 3D polylines between modern lightweight and classic polylines (heavyweight) definitions.

Shape Lite Pro Mechanical BIM

8.54.1 Method

The main benefit of this command is to reduce the drawing file size by using lightweight polyline definitions.

This command does not convert the following types of polylines:

- Curve-fit.
- Splined polylines.
- Polylines with extended object data stored on their vertices.

Note: BricsCAD ignores ineligible entities.

You may not need to use this command in the following situations:

- When editing commands apply to both classic and lightweight polylines.
- You can also use the PLINETYPE system variable to specify if classic polylines are automatically converted to lightweight polylines when an older drawing is opened. This system variable determines which style of polyline is created in new drawings.

Note: You can enter this command transparently during commands ('convertpoly).

8.54.2 Options within the command

Heavy
Converts lightweight and 3D polylines to heavyweight ones.
This might be necessary to make drawings compatible with certain software.

Light
Converts heavyweight and 3D polylines to lightweight ones.

Split 3d polyline into planar polylines if it is not planar?
Toggles between **Yes** and **No**.

3dPoly
Converts lightweight and heavyweight polylines to 3D polylines.

8.55 CONVERTPSTYLES command

Converts the current drawing from Color Dependent (CTB) to Named (STB) plot style mode and vice versa.

Shape Lite Pro Mechanical BIM

Note: A drawing can use either CTB or STB plot styles, but not both.

8.55.1 Description

Converts the drawing's plot style tables before converting the drawing, using the CONVERTCTB command.

8.56 CONVTOMESH command

Converts entities to mesh entities.

Shape Lite Pro Mechanical BIM

8.56.1 Description

Converts valid 2D and 3D entities to Mesh entities. Valid entities include:

- 3D Solid
- Surface
- Polygon Mesh
- Region
- Closed Polyline

8.56.2 Method

Select a valid entity then select another valid entity or press Enter to complete the command.

Note:

- The BIM data and GUID are maintained after converting.
- The value of the FACETRES system variable influences the display resolution of a mesh.

8.57 CONVTOSOLID command

Converts entities to solid entities.

Shape Lite Pro Mechanical BIM

8.57.1 Description

Converts valid 2D and 3D entities to 3D Solid entities. Valid entities include:

- Watertight 3D Mesh
- Watertight 3D Surface
- 3D Polygon Mesh
- 3D Polyface Mesh
- Circle with thickness
- Closed Polyline of non-zero thickness and uniform width

8.57.2 Method

Select a valid entity then select another valid entity or press Enter to complete the command.

Note: The BIM data and GUID are maintained after converting.

8.58 CONVTOSURFACE command

Converts entities to Surface entities.

Shape Lite Pro Mechanical BIM

8.58.1 Description

Converts valid 2D and 3D entities to Surface entities. Valid entities include:

- 2D Solid
- 3D Solid
- Region
- Open Polyline with non-zero thickness and zero width
- Line with non-zero thickness
- Arc with non-zero thickness
- Circle with non-zero thickness
- Mesh

- Planar 2D Face

8.58.2 Method

Select a valid entity then select another valid entity or press Enter to complete the command.

Note: The BIM data and GUID are maintained after converting.

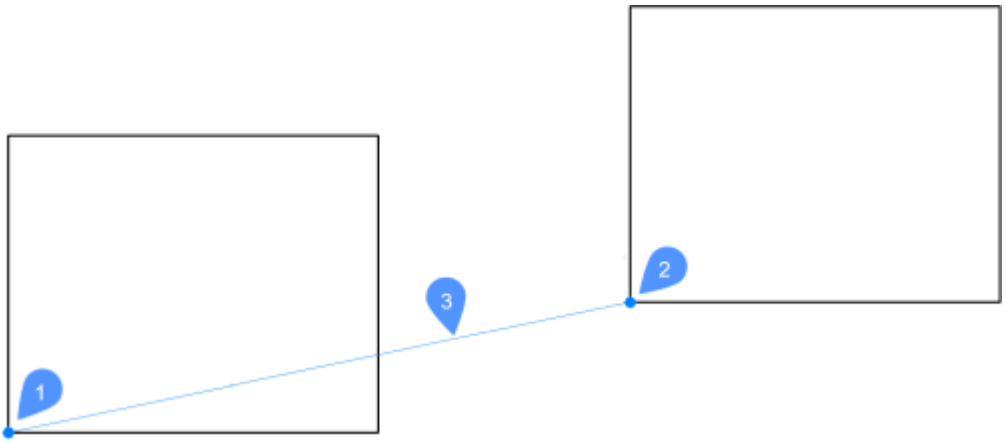
8.59 COPY command

Makes copies of entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: CO, CP



1 Base point

2 Second point

3 Displacement distance

8.59.1 Method

This command allows you to make a single or multiple copies of entities by entering a base point and a displacement vector.

8.59.2 Options within the command

Displacement

Specify the displacement vector (the distance at which to position the copy). 'Vector' means that you specify the distance and angle at the same time.

Note: When Dynamic Input Mode is active, you can type a distance and angle in the dynamic input fields.

mOde

Toggle between single and multiple copying modes.

Array
Specifies the number of copies to create and the distance between each copy or the distance between the first and last copies.

Multiple
In single-copy mode only: Switches to multiple-copy mode.

Undo
In multiple-copy mode only: Undoes the last copy operation.

Repeat
Repeats the copy using the same displacement.

Exit
In multiple-copy mode only: Exits this command.

8.60 COPYBASE command

Copies entities to the Clipboard.

Shape Lite Pro Mechanical BIM

Icon: 

8.60.1 Description

Copies entities to the Clipboard with a base point for pasting into the same drawing or another drawing.

Note: When you paste entities copied with the COPYBASE command, it uses a user-defined base point.

8.61 COPYCLIP command

Copies entities to the Clipboard.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The PICTUREEXPORTSCALE system variable sets the resolution of the image when the exported geometry is pasted in bitmap format, such as in a Word document.

8.61.1 Description

Copies selected entities to the Clipboard for pasting into drawings and other documents.

8.62 COPYEDATA command

Copies extended entity data from one entity to others.

Shape Lite Pro Mechanical BIM

Icon: 

8.62.1 Method

Enter the name of the application to which the entity data belongs, select the entity from which to copy Entity Data and one or more entities to which to copy Entity Data (that will receive the edata).

Extended Entity Data can be created with EDITEDATA command.

8.62.2 Options

? to list application names
Lists the names of applications loaded in the current drawing.

8.63 COPYGUIDED command

Copies entities using guide curves.

Shape Lite Pro Mechanical BIM

Icon:

8.63.1 Description

Automatically aligns copied entities with relevant geometry, using temporary guide curves. Polyline segments and multiline segments are also accepted as guide lines.

8.63.2 Method

The command can be executed in two ways:

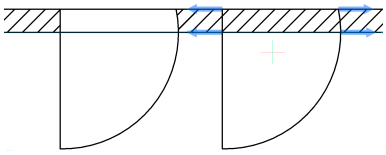
Pre-pick mode
First select entities, then launch the command.

Note: The entities to be copied are displayed in green.

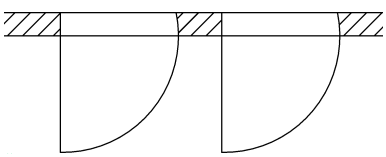
Post-pick mode
Launch the command, then select the entities using a selection window.

Note: All entities that fall completely within the selection window, including polyline segments, are included in the copy selection set and are displayed in yellow. Entities that cross the selection window are used as guide curves and are displayed in blue.

Blue arrows indicate the anchor points and direction of the guide curves. The copied entities will only align with geometry that matches the number of guide curves and distances between them.



Click to place the copy or enter a distance in the dynamic entry fields.



8.63.3 Options within the command

Polygonal
Creates a polygonal selection window.

Rectangular
Creates a rectangular selection window.

Region
Pastes the clipped area inside the selection window.

Entities
Pastes the entities inside the selection window.

Save detail
Allows to save a detail as a block/Parametric block.

Note: Displays the **Write block to file** dialog box. See the related article **Write block to file dialog box**.

Flip entities
Flips the entities along a line perpendicular to the reference curves.

Note: You can also press the Shift key to flip the entities.

choose base Point
Defines a base point for insertion.

use Center point
Uses the center point for insertion.

switch to 3D mode
This option links the COPYGUIDED command to the COPYGUIDED3D functionality.
See the COPYGUIDED3D command for more information.

Manually specify source faces
Sets the source faces manually.

Reset
Resets the selection entirely or alters the automatic selection.

Automatic detection of source faces
Sets the recognition of source faces to automatic.

specify Base point
Allows you to pick a point or enter coordinates to specify the base point.

Rotate detail
Allows you to rotate the detail dynamically.

Disable display of source faces
Toggles the display of source faces.

switch back to 2D mode
Switches back to 2D mode, if 3D mode was previously selected.

8.64 COPYGUIDED3D command

Copies 3D Solids, Block references or a set of faces from one location to another, using reference faces of the user's choice.

Shape Lite Pro Mechanical BIM

8.64.1 Method

Selecting a solid, a block or a set of faces activates the automatic detection of connections and source faces. These source faces should not intersect with the detail extents.

Only faces with analytical geometry (planar, cylindrical, spherical, conical and toroidal) are taken into account.

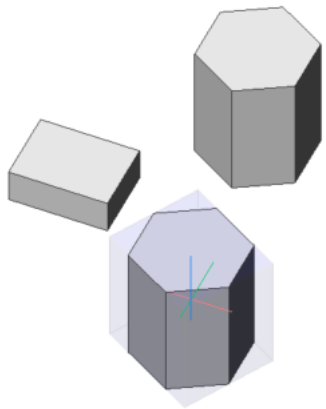
To find insertion points, the algorithm uses faces in a similar way as how COPYGUIDED uses curves.

Dynamic dimensions are used to position the detail entities.

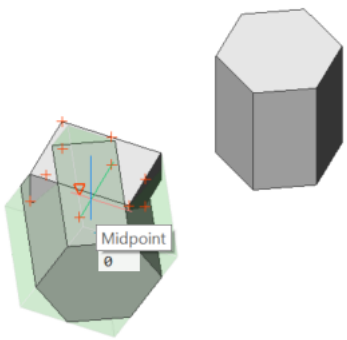
The detail can transform to fit a new position. The transformation of the detail can consist of translations, rotations and mirrorings.

The detail can be visualized in two colors:

- Blue means the detail extent is in **Free mode** without any possibilities to insert.



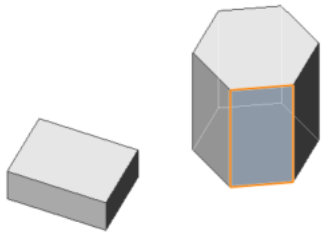
- Green means the algorithm has recognized a placement for the detail extents, the transformation has been found and the detailed has already been transformed in the preview. Therefore, the next mouse-click will insert the detail using the found transformation.
-



When a possible insertion has been detected, press the **Ctrl** key to cycle through all detected alternatives for the current cursor placement. Next, hold down the **Shift** key to fix a set of faces, to prevent the command from searching for these faces.

8.64.2 Option within the command

Manually specify source faces
Sets the source faces manually.



Reset
Resets the selection entirely or alters the automatic selection.

Automatic detection of source faces
Sets the recognition of source faces to automatic.

specify Base point
Allows you to pick a point or enter coordinates to specify the base point.

Rotate detail
Allows you to rotate the detail dynamically.

Disable display of source faces
Toggles the display of source faces. When this option is Off, the **Enable display of source faces** becomes available.

8.65 COPYHIST command

Copies all text from the Command line history to the Clipboard.

Shape Lite Pro Mechanical BIM

Note: The SCRLHIST system variable determines how many lines of command history the Prompt History window retains.

8.66 COPYM command (Express Tools)

Makes multiple copies of selected entities.

Shape Lite Pro Mechanical BIM

Icon:

8.66.1 Method

Select the entities you want to copy, then specify the base point.

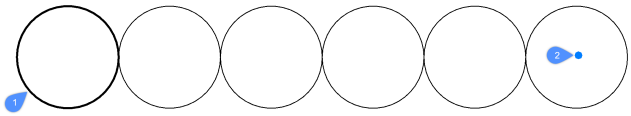
8.66.2 Options within the command

Repeat

The last copy is repeated, using the same offset distance and direction.

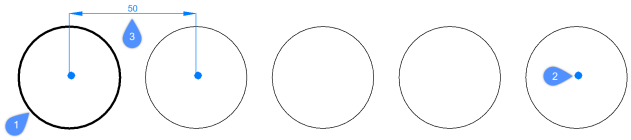
Divide

Creates a number of copies of the original entity (1), based on an end point for division (2) and a number of copies.



Measure

Creates a number of copies of the original entity (1), based on an end point for measure (2) and a distance between copies (3).

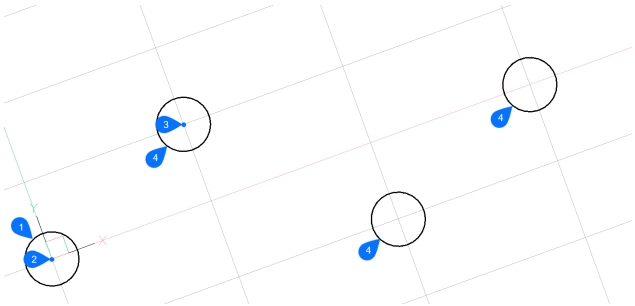


Array

Allows you to choose between the **Pick**, **Measure** and **Divide** options.

Pick

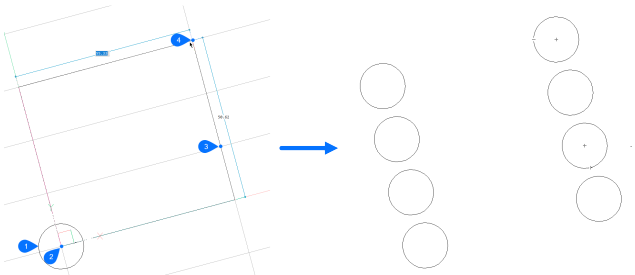
Allows you to interactively pick locations for the copied entities, using a temporary array.



After selecting an object to copy (1) and a base point (2), specify an angle and define the column and row spacing for the array by selecting a point (3). The cursor snaps to the intersection of each column and row, so you can only pick one array element at a time (4).

Measure

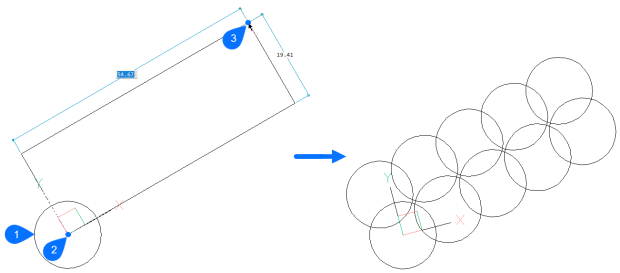
Allows you to specify an array set to be filled with copies of the selected entity based on a specified inter-distance.



After selecting an object to copy (1) and a base point (2), specify an angle and define the column and row spacing for the array by selecting a second point (3). Next, specify the third point (4) to indicate the array zone.

Divide

Allows you to define a box that represents an array set, to be filled with a specified number of copies of the selected entity.



Specify an angle and choose the other corner for the array set, then specify the number of columns and rows.

8.67 COPYTOLAYER command (Express Tools)

Copies entities to another layer.

Shape Lite Pro Mechanical BIM

8.67.1 Description

Creates duplicates of selected entities on a layer specified by the user. You can specify a different location for the duplicated entities.

8.67.2 Method

There are two methods to copy to layer:

- Select entity on target layer.
- Select target layer in the **Copy To Layer** dialog box.

8.67.3 Options within the command

Name

Displays the **Copy To Layer** dialog box that allows selecting a target layer. For more information, see related article **Copy To Layer dialog box**

Displacement

Specify the displacement vector (the distance at which to position the copy). **Vector** means that you specify the distance and angle at the same time.

Note: When **Dynamic Input Mode** is active, you can type a distance and angle in the dynamic input fields.

8.68 -COPYTOLAYER command (Express Tools)

Copies entities to another layer.

Shape Lite Pro Mechanical BIM

8.68.1 Description

Creates duplicates of selected entities on a specified layer. You can specify a different location for the duplicated entities.

8.68.2 Options within the command

?

Lists the available layer names.

=

Allows you to select an entity with the necessary layer name.

return

Creates the copies in the same location as the selected entities.

8.69 CORRIDOR command

Creates a corridor from a corridor template.

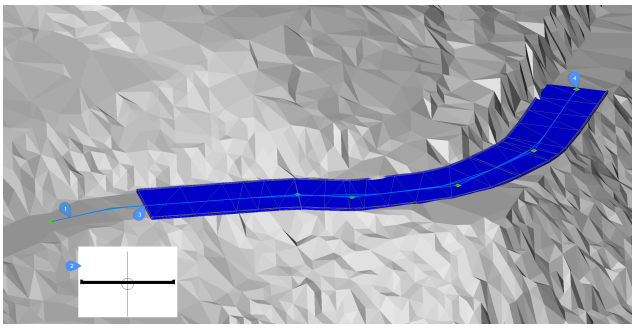
Shape Lite Pro Mechanical BIM

Icon:

8.69.1 Methods

There are two methods for creating a corridor:

1 **Creates a corridor using a corridor template along an 3D alignment line.**
   - Select 3d alignment used as a corridor baseline (1).
   - Select corridor template (2).
   - Enter region start station (the starting point along the 3D alignment line) (3).
   - Enter region end station (the ending point along the 3D alignment line) (4).



2 **Creates a simple corridor using a corridor template along a path defined by picking horizontal alignment PI points.**
   - Select TIN Surface
   - Select corridor template
   - Pick horizontal alignment PI point: pick as many horizontal PI points needed and press Enter to

finish.

8.69.2 Options within the command

create Simple corridor
Creates a simple corridor using a corridor template along a path defined by picking horizontal alignment PI points.

Predefined
Sets the predefined corridor template and specifies the template position.

Change template
Allows to change the corridor template.

8.70 CORRIDOREDIT command

Adds or removes regions for a selected corridor.

Shape Lite Pro Mechanical BIM

Icon: 

8.70.1 Method

The CORRIDOREDIT command can be used to:

- Add or remove baselines for a selected corridor.
- Add, remove, split or merge regions for a selected corridor.
- Add stations to a selected corridor.

8.70.2 Options within the command

Add Baseline
Adds a new corridor baseline.

Remove Baseline
Removes a corridor baseline.

Add Region
Adds a new region to a selected corridor baseline.

Enter region start station
Allows you to enter the starting point along the 3D alignment line.

Enter region end station
Allows you to enter the ending point along the 3D alignment line.

Note: The available intervals are displayed in the command window.

Remove Region
Removes a selected region.

whole Baseline
Allows you to select the whole baseline.

by Index
Allows you to enter an index which corresponds to the baseline/region.

Note: For regions, the index is calculated from region start station to region end station.

by Name
Allows you to select a baseline/region by its name.

by Range
Allows you to select a range baseline/region.

Select region range start
Allows you to indicate the start region.

Select region range end
Allows you to indicate the ending region.

Note: The available options are displayed in the command window.

Add Station
Adds station to a region.

Enter region start station
Allows you to enter the starting point along the 3D alignment line.

Enter region end station
Allows you to enter the ending point along the 3D alignment line.

Note: The available intervals are displayed in the command window.

Split Region
Splits a selected region.

Merge Region
Merges regions.

8.71 CORRIDOREXTRACT command

Extracts a 3D solid, 3D mesh, TIN surface, 3D polyline or outer boundaries from a corridor.

Shape Lite Pro Mechanical BIM

Icon: 

8.71.1 Description

Extracts a 3D solid, 3D mesh, TIN surface, 3D polyline or outer boundaries from a corridor. Corridor template element (CTE) is defined by points, links and shapes and each of these can contain multiple codes. Corridor region applies template (a collection of CTEs) at incremental stations along it's baseline. Subsequent points, links or shapes are stitched together to form a newly extracted entity. Points are used to extract 3D polylines and outer boundaries for each region separately. Similarly shapes are used to extract 3D solids or 3D meshes in each region. Whereas links are used to create TIN surfaces, but they are always combined from all regions into a single TIN surface, and can only be specified using codes.

8.71.2 Method

There are three methods to use CORRIDOREXTRACT command:

- Extract all 3D solids, 3D meshes, 3D polylines, TIN surface or outer boundaries.
- Extract individual 3D solids, 3D meshes, 3D polylines TIN surface or outer boundaries.
- Extract 3D solids, 3D meshes, 3D polylines, TIN surface or outer boundaries by code.

8.71.3 Options within the command

Mesh
Extracts a 3D mesh.

All Shapes
Extracts 3D mesh or 3D solid from all shapes.

by Codes
Extracts 3D mesh or 3D solid by codes.

Solid
Extracts a 3D solid.

Tin
Specifies the links to extract.

? to list all link codes
Lists all link codes in command line.

liSt selected link codes
Lists the selected link codes.

STrings
Extracts 3D polylines.

All Stringlines
Extracts a 3D polyline from all stringlines.

Note: Stringlines are lines that connect points with the same code value along the corridor baseline.

by Codes
Extracts a 3D polyline from all stringlines by code.

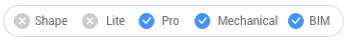
Polylines
Extracts a 3D polyline.

outer Boundaries
Extracts the outer boundary as a 3D polyline.

Create dynamic link
Specifies if the created boundary or stringline should be linked to the original corridor.

8.72 CORRIDORTEMPLATE command

Creates a corridor template.



Icon:

8.72.1 Method

Pick a point to create a corridor template.

Add corridor template elements to existing corridor template.

8.72.2 Options within the command

Add template element
Select a corridor template element that you wish to append.

8.73 CORRIDORTEMPLATEELEMENT command

Creates a corridor template element from polylines with a specified base point and orientation (left, right, none).

Shape Lite Pro Mechanical BIM

Icon:

8.73.1 Description

The geometry of the corridor template element is created on the basis of the specified polylines, which can later be edited with the command CORRIDORTEMPLATEELEMENTEDIT.

Input polylines can contain a large number of segments. An individual segment can be drawn as a line or arc. The CVARCTESSELLATIONTEMPLATEELEMENT system variable controls the Mid-Ordinate Distance parameter which approximates arc segments.

8.73.2 Method

Select polylines to create a corridor template element and specify the base point.

8.73.3 Options within the command

None
No orientation.

Right
Right orientation.

Left
Left orientation.

8.74 CORRIDORTEMPLATEELEMENTEDIT command

Edits corridor template elements by adding or removing components and targets.

Shape Lite Pro Mechanical BIM

Icons:

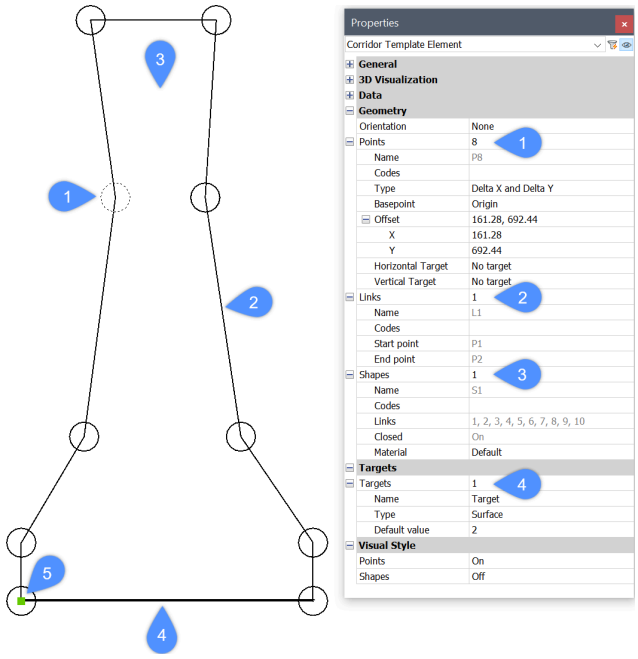


8.74.1 Description

Edits corridor template elements by adding or removing components (points, links, shapes) and targets to an existing template element.

Note:

- After insertion, points (1), links (2), shapes (3) and targets (4) can be also edited in the Properties panel.
- The currently selected point and link are highlighted.
- The green square (5) represents the origin of the corridor template element.



8.74.2 Options within the command

Add Point

Adds a new point to an existing template element by specifying its location.

Basepoint

Allows to specify a basepoint, relative to which the position of the new point is determined. A new point can be inserted directly after the specified basepoint or as the last point of the selected template element.

by Index

Allows to enter an index which corresponds to the point template element component (point, link, shape) by the index. The index represents the sequence number of the template element component.

Point P1 has an index value of 1, point P2 value has an index value of 2, and so on. Link L1 has an index value of 1, link L2 has an index value of 2, and so on. The same applies to shapes.

Add new points directly after the basepoint?

Toggles between Yes or No.

Unset basepoint

Unsets basepoint, which is set to the origin for the new point.

unset insertion Index

Unsets insertion index, which is set to the last value for the new point.

Let's take an example from the figure above, where the template element consists of 10 points, named P1 to P10. Assume that we want to insert a new point directly after the specified basepoint P5. If we select the **unset insertion Index** option in the command line, the new point will get an index value equal to 11, so the name of new point will be P11. If this option is not selected, the new point will get an index value of 6 and the point name will be P6. Existing points from P6 onwards are re-indexed and renamed.

insert bEfore

Adds new point before a specified point.

If a new point is added to the template element before the existing point P5, the new point will get an index value equal to 5, and the name P5. The existing point will be re-indexed and renamed to P6.

Links between existing points and a newly added point are not created automatically, so these points need to be re-linked.

insert After

Adds new point after a specified point. The index of a newly added point is one unit higher than the index of an existing selected point.

If a new point is added to the template element after the existing point P5, the new point will get an index value equal to 6, and the name P6. Existing points from P6 onwards are re-indexed and renamed.

Links between existing points and a newly added point are not created automatically, so these points need to be re-linked.

Remove Point

Removes the template element point by clicking on it or by typing an index. The remaining points are re-indexed accordingly after editing.

Add Link

Creates a link between selected points.

Remove Link

Removes the template element link by clicking on it or by typing an index. The remaining links are re-indexed accordingly after editing.

Add Shape

Creates shapes from template element links.

Shapes determine the geometry of the cross-sectional elements. They can be created from a single template element link or from multiple links forming a closed polygon. We assign materials to the shapes in the Properties panel for a realistic representation of the Corridor model.

Remove Shape

Remove the template element shape by clicking on it or by typing an index. The remaining shapes are re-indexed accordingly after editing.

Add Target

Allows you to create **Surface**, **Horizontal** and **Vertical** targets.

When the geometry of an individual corridor template element must follow a specific surface, horizontal or vertical object, a target is created and added to the corresponding point on the corridor template element.

Remove Target

Removes template element target by clicking on it or by typing an index.

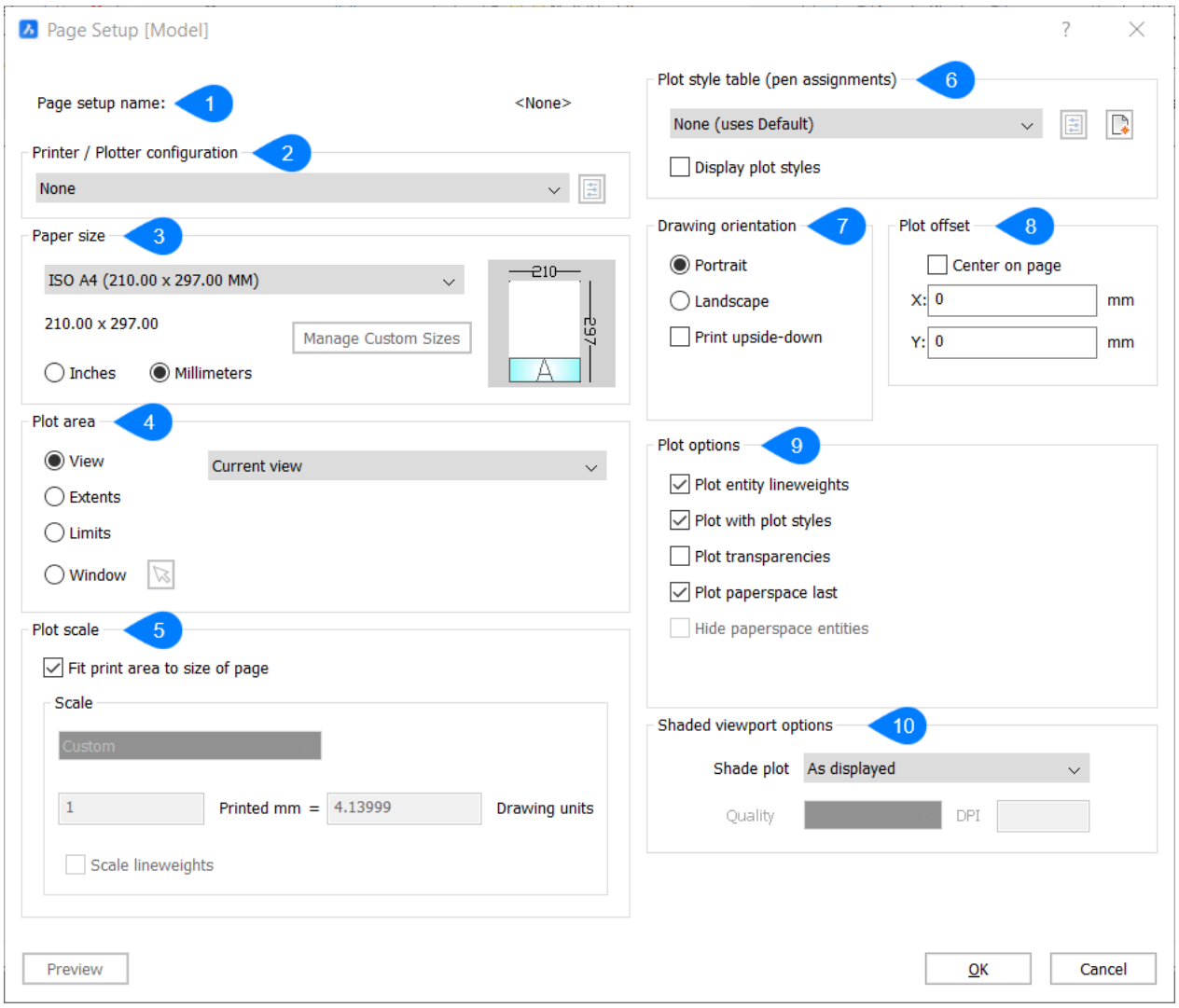
8.75 CPAGESETUP command

Opens the **Page Setup** dialog box.

Shape Lite Pro Mechanical BIM

8.75.1 Description

Opens the **Page Setup** dialog box to view and modify the page setup for the current layout or model space.



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

8.75.2 Page setup name

Displays the name of the current setup.

8.75.3 Printer / Plotter configuration

Selects the printer or plotter.

8.75.4 Paper size

Specifies the size of paper by selecting a standard size from the drop-down list. These are sizes supported by the printer.

Note: The size can be displayed in inches or millimeters by checking one of the two options.

8.75.5 Plot area

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.

Note: 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.

8.75.6 Plot scale

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.

8.75.7 Plot style table

Specifies the plot style table to use for the printed output, which assigns properties to "pens," colors, and entities.

8.75.8 Drawing orientation

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.

8.75.9 Plot offset

Specifies the offset distance for the print.

Note: 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.

8.75.10 Plot options

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.

Note: This option is only available when printing layouts.

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

Note: This option is disabled when printing model space.

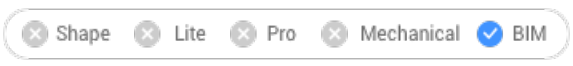
8.75.11 Shaded viewport options

Override the visual style of the current view when printing model space.

Shade plot
Choose a shade style to plot the drawing with.

8.76 CREATEBIMPROJECT command

Opens the **New Project** dialog box.



8.76.1 Description

Opens the **New Project** dialog box, which allows you to easily create a new BIM Project and specify the project name, model name and path of the new BIM project.

8.77 CREATELIBRARYBLOCK command

Creates and categorizes components.

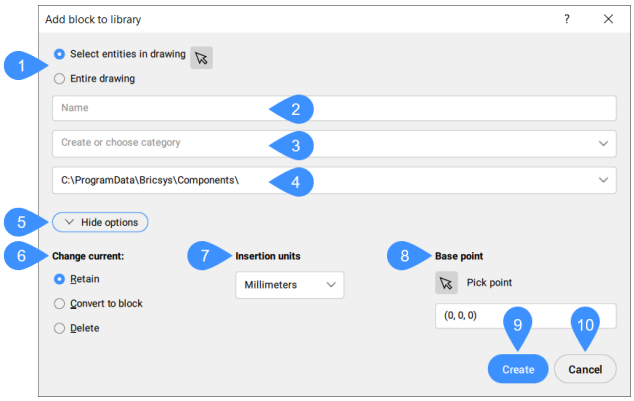
Shape Lite Pro Mechanical BIM

Icon:

8.77.1 Description

Opens the **Add block to library** dialog box.

The **Add block to library** dialog box creates a block definition and adds it to the block 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

8.77.2 Selection set

Specifies which entities are included in the block definition. You can select entities in the drawing or use the entire drawing.

8.77.3 Name

Specifies the name of the block.

8.77.4 Category

Specifies a category for the block. You can choose from a list of existing categories or enter a new category name.

8.77.5 Location

Specifies the location where the block DWG file is saved. Available locations are controlled by the **Library directory path** setting (COMPONENTSPATH system variable).

8.77.6 Show/Hide Options

Shows or hides additional options. The available options depend on the selection set.

8.77.7 Change current

Specifies 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.

8.77.8 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.

8.77.9 Create

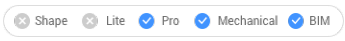
Creates a DWG file for the block definition in the specified location, adds the block to the **Library** panel under the specified category and closes the dialog box.

8.77.10 Cancel

Closes the dialog box without creating the block definition.

8.78 -CREATELIBRARYBLOCK command

Creates and categorizes components via Command line.



8.78.1 Description

Adds entities to the **Library** panel in the form of components that are categorized and displayed as thumbnails. This command is meant for macros.

The component will be added to an existing category or to a new category. If a specific category is open in the **Library** panel, the command will display the option to save in this category or a new category.

Categories are listed in alphabetical order:

- 1 - Annotation 2d
- 2 - Bathroom
- 3 - Bathroom 2d
- 4 - Bedroom
- 5 - Bedroom 2d
- 6 - Dining room
- 7 - Dining room 2d
- 8 - Doors
- 9 - Doors advanced
- 10 - Fire protection 2d
- 11 - Fixtures and fittings
- 12 - Floor plan 2d

- 13 - Garden 2d
- 14 - Holes
- 15 - Kitchen
- 16 - Kitchen 2d
- 17 - Lighting
- 18 - Living room
- 19 - Living room 2d
- 20 - Mep flow connection points
- 21 - Office
- 22 - Office 2d
- 23 - Outdoors
- 24 - People
- 25 - Roof
- 26 - Sheet metal
- 27 - Standard parts
- 28 - Tools
- 29 - Transport
- 30 - Transport 2d
- 31 - Vertical mobility
- 32 - Windows
- 33 - Windows advanced
- 0 - Add a new category
- <rootCat (current category)> - add component to the current category

A thumbnail is created from the desired view:

- TFL - Top front left isometric view
- TFR - Top front right isometric view
- TBL - Top back left isometric view
- TBR - Top back right isometric view
- T - Top view

8.79 CREATETHUMBNAIL command

Creates a thumbnail image of the current drawing.

Shape Lite Pro Mechanical BIM

8.79.1 Description

Creates custom thumbnail views to show the content of drawing files by file managers and other non-CAD programs. The thumbnails are small raster preview images, and usually are created automatically of the current view as the drawing is saved.

8.79.2 Method

Accept the thumbnail frame by zooming and panning the drawing to the position in which you prefer to save the thumbnail image. Opens the **Save Drawing As** dialog box to save the thumbnail image.

8.80 CUILOAD command

Opens the **Customization Groups** dialog box.

Shape Lite Pro Mechanical BIM

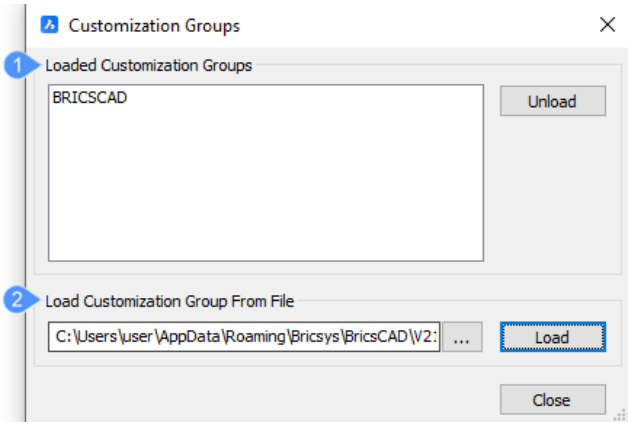
8.80.1 Description

Opens the **Customization Groups** dialog box to load and unload customization groups.

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\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*).

When you launch BricsCAD the first time, the classic-ribbon.cui file is loaded automatically.



1 Loaded Customization Groups

2 Load Customization Group from File

8.80.2 Loaded Customization Groups

A list with the names of the customization files already loaded displays.

To unload a CUI file, select it from the list and click **Unload**.

8.80.3 Load Customization Group from File

Click the browse button to select a partial CUI file and click **Load** to load it.

Note: There are three standard CUI files available by default:

- Modern

- Classic-ribbon
- Classic-toolbars

8.81 CUIUNLOAD command

Opens the **Customization Groups** dialog box.

Shape Lite Pro Mechanical BIM

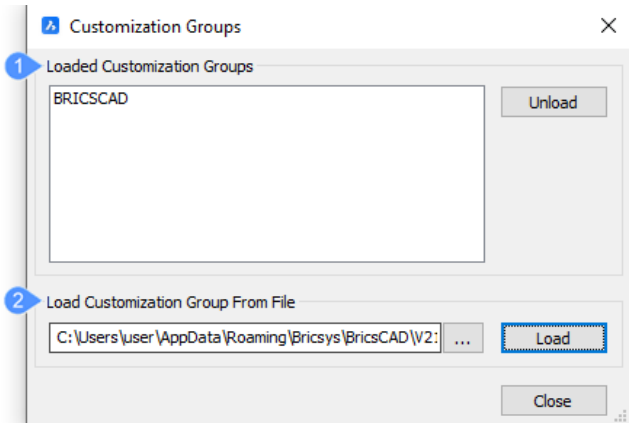
8.81.1 Description

Opens the **Customization Groups** dialog box to load and unload customization groups.

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\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*).

When you launch BricsCAD the first time, the classic-ribbon.cui file is loaded automatically.



1 Loaded Customization Groups

2 Load Customization Group from File

8.81.2 Loaded Customization Groups

A list with the names of the customization files already loaded displays.

To unload a CUI file, select it from the list and click **Unload**.

8.81.3 Load Customization Group from File

Click the browse button to select a partial CUI file and click **Load** to load it.

Note: There are three standard CUI files available by default:

- Modern
- Classic-ribbon
- Classic-toolbars

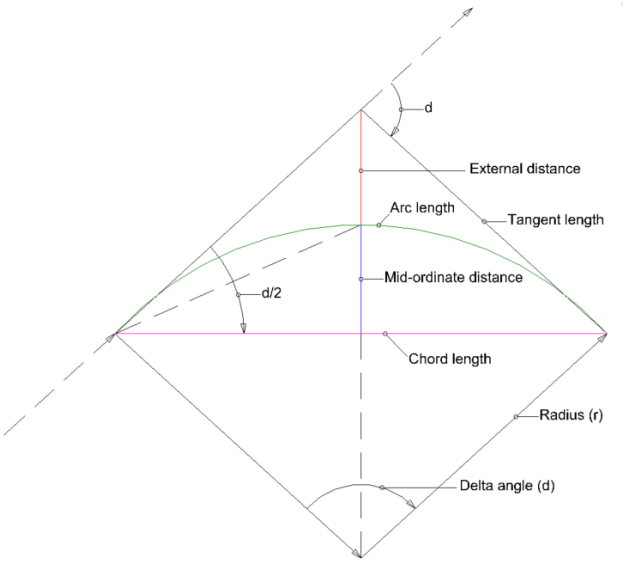
8.82 CURVECALC command

Calculates curve parameters based on input and adds it to the drawing.

8.82.1 Description

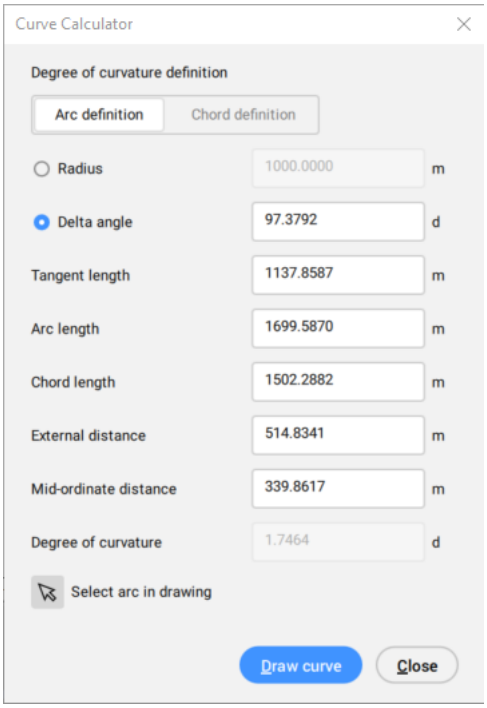
Opens the **Curve Calculator** dialog box to set the curve parameters and draw the curve in the drawing.

The curve parameters are presented in the following picture:



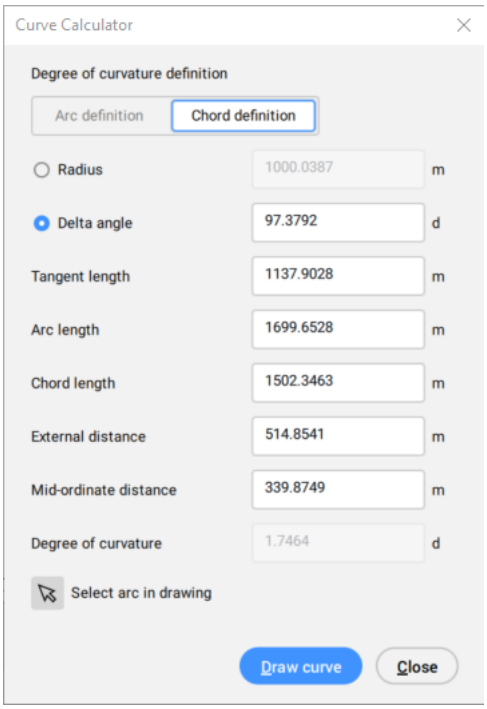
8.82.2 Arc definition

The **Degree of curvature** defined by the **Arc definition** means that the arc determines the degree of curvature. This definition is commonly used for roadway curves, where the degree of the curve corresponds to an angle of an arc of 100 units.



8.82.3 Chord definition

The **Degree of curvature** defined by the **Chord definition** means that the chord determines the degree of curvature. This definition is commonly used for railway curves, where the degree of the curve corresponds to a chord of 100 units.



The calculation is performed based on the **Radius** or the **Delta angle**.

Radius
If checked, the calculation starts from a fixed radius.

Delta angle
If checked, the calculation starts from a fixed delta angle.

Tangent length
Sets the tangent length of the curve.

Arc length
Sets the arc length of the curve.

Chord length
Sets the chord length of the curve.

External distance
Sets the external length of the curve.

Mid-ordinate distance
Sets the mid-ordinate length of the curve.

Degree of curvature
Sets the degree of curvature. Based on **Arc definition**, the degree of curvature is determined by the arc, and based on **Chord definition**, the degree of curvature is determined by the chord.

Select arc in drawing
Allows you to select an arc from the drawing.

Draw curve
Temporarily dismisses the **Curve Calculator** dialog box to draw the calculated curve in the drawing.

8.82.4 Options within the command

Select Entity
Allows you to select an entity from the drawing.

Note: When you choose an entity as the starting point of the curve, the curve drawn will be tangent to that entity.

Point
Allows you to select the curve origin by picking a point in the drawing.

Select curve direction
Allows you to set the curve direction.

Select curve side
Allows you to pick the side of the curve.

Left
Places the curve to the left side.

Right
Places the curve to the right side.

8.83 CUSTOMIZE command

Opens the **Customize** dialog box.

Shape Lite Pro Mechanical BIM

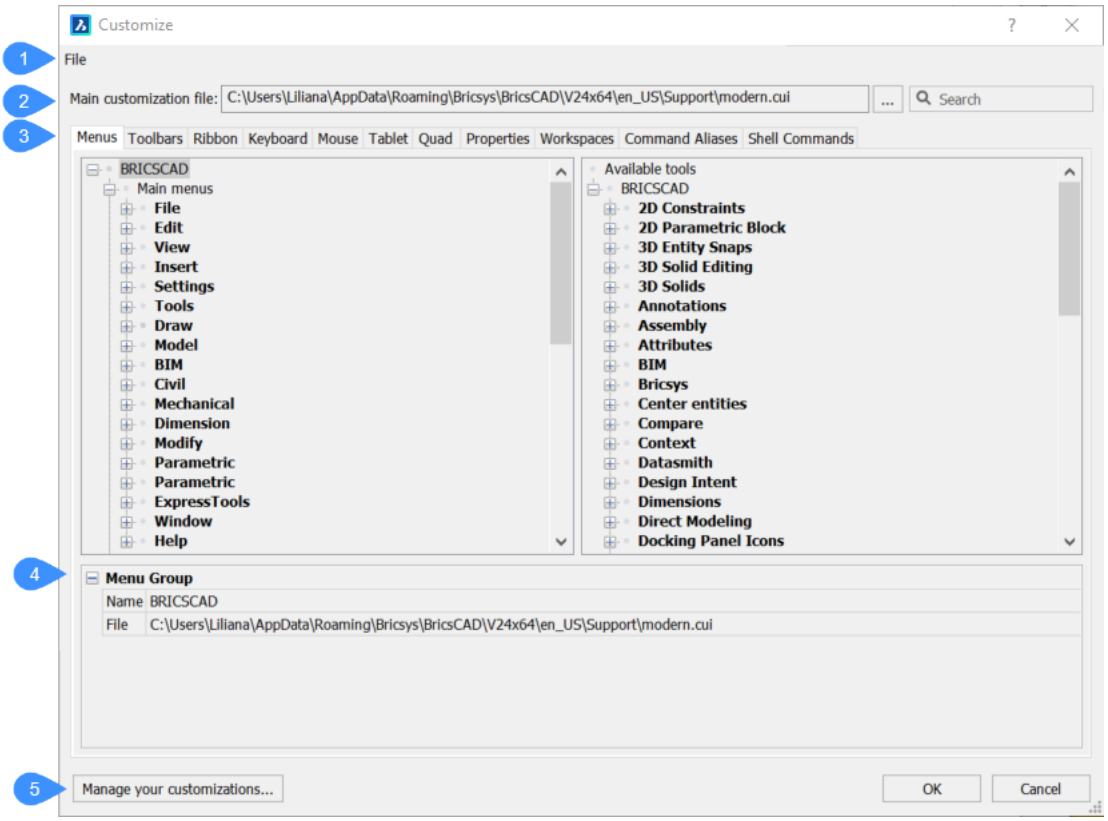
Icon: 

Alias: CUI

8.83.1 Description

Opens the **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

8.83.2 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.

Note: 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.

Note: 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.

8.83.3 Main customization file

Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface.

Note: 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.

Note: There are three standard CUI files available by default:

- Modern
- Classic-ribbon
- Classic-toolbars

Search
Searches the Available Tools list for command names.

8.83.4 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.

Note: 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 or menu item.

Title
Specifies the name that is displayed by the menu, menu item or submenu.

Note: 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. 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 item or submenu.

Tool ID
Identifies the menu item or sub menu 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 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.

Note: 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/Toolbar Button
Displays the options of the selected toolbar, toolbar button, toolbar control or toolbar flyout.

Title
Specifies the name that is displayed by the tooltip.

ID
Unique identifier for each toolbar 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
Specifies the position of the selected toolbar. You can choose between **Floating**, **Top**, **Left**, **Bottom**, **Right**.

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 toolbar.

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.

Note: If you select **Add launcher**, **Insert toggle button** or **Insert command button** from the context menu, the **Add ribbon dialog launcher**, **Add ribbon toggle button** or **Add ribbon command button** dialog boxes display:

- **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 **Ribbon Panel Reference**.

ID
Identifies the item in the CUI file. (The ID is assigned by BricsCAD).

Collapse
Allows you to choose if the panel will collapse automatically or if it will never collapse.

Label
Specifies the name that is displayed by the selected **Ribbon Tab** or **Ribbon Panel Reference**.

Title
Specifies the name of the selected **Ribbon Tab** or **Ribbon Panel Reference**.

Key Tip

Note: Ribbon keytips are not yet implemented in BricsCAD.

Menu Group/Ribbon Panel/Split Button/Ribbon Row Panel/Command Button/Toggle Button
Displays the options of the selected **Ribbon Panel**, **Split Button**, **Ribbon Row Panel**, **Command Button** or **Toggle Button**.

ID
Identifies the item in the CUI file. The ID is assigned by BricsCAD.

Label
Specifies the name that is displayed by the selected Ribbon item.

Title
Specifies the name of the selected **Ribbon Panel**.

Button Style
Specifies the display mode for the selected button. You can choose between **Small with text**, **Small without text**, **Large with text (vertical)**, **Large with text (horizontal)** and **Large without text**.

Note: The icon size for the small options is 16×16 pixels, while for the large options is 32×32 pixels.

Behavior
Determines how the topmost button behaves when you click on it. You can choose between **Drop down**, **Drop down with recent**, **Split**, **Split with recent**, **Split with recent (static text)**. The options determine whether the button displays the default command (the first one in the list of buttons), or the most recently used one.

Note: Split buttons let you click the upper half to execute the most-recently-used command, or lower half to display the drop-list (flyout).

List Style

Note: This option is not yet implemented in BricsCAD.

Grouping

Note: This option is not yet implemented in BricsCAD.

Image
Specifies the image to display for the selected Ribbon item.

Resize Style

Note: This option is not yet implemented in BricsCAD.

Resize Priority

Note: This option is not yet implemented in BricsCAD.

Justify Top

Note: This option is not yet implemented in BricsCAD.

Tool ID
Identifies the ribbon item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help
Specifies the help string displayed on the Ribbon.

Command
Specifies the command(s) or macros.

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.

Note: 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 keyboard shortcut.

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.

Note: 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 button item to other elements in the CUI file. The Tool ID is assigned by BricsCAD.

Title
Specifies the name that is displayed by the button item.

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 button item. 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.

Note: If you select **Append button** or **Insert button** from the context menu, the **Add button item** dialog box displays.

Menu Group/Button Group
Displays the options of the selected 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.

Note: 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 Quad.

Command
Specifies the command(s) or macros related to the Quad button.

Image
Specifies the image to display for the Quad 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.

Note: 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.

Name
Shows the name of the workspace displayed in the workspace list in the status bar and by the Workspace toolbar (reported by the WSCURRENT system variable).

Display
Determines if the workspace name is displayed by the status bar and toolbar drop lists.

Description
Help-like description displayed in the status bar.

ID
Identifies the workspace item in the CUI file. The ID is assigned by BricsCAD.

Menu Bar
Toggles the display state of the menu bar. See also MENUBAR (EXCEPT OS X) system variable.

Scrollbars
Toggles the display of scroll bars. See also the WNDLSCRL system variable.

Default
Toggles whether this workspace is the default one when BricsCAD starts.

- **Yes**: shows this workspace when BricsCAD starts.

- **No**: does not show this workspace.

Stack Type
Determines how panels are displayed. See also the STACKPANELTYPE system variable.

Panel Button Size
Specifies the initial size of buttons on panels.

Quad Button Size
Specifies the initial size of buttons on Quad.

Ribbon Tool Size
Specifies the initial size of buttons on Ribbon.

Tool Button Size
Specifies the initial size of buttons on tools.

Note:

- **Small**: displays 16×16 icons.
- **Large**: displays 24×24 icons.
- **Extra large**: displays 32×32 icons.

Ribbon Panel Margin
Specifies the size of blank space at ribbon panel edges, measured in pixels. See also the RIBBONPANELMARGIN system variable.

Toolbar Margin
Specifies margin above and below each toolbar, measured in pixels. See also the TOOLBARMARGIN system variable.

Tool Icon Padding
Specifies margin between icons on each toolbar, measured in pixels.

Delete Tool
Specifies what to do with tool entities following the SUBTRACT command. See also the DELETETOOL system variable.

DMAUDIT Detail Level
Specifies messages to display. See also the DMAUDITLEVEL system variable.

DMPUSHPULL Subtract
Specifies whether to enable subtract mode in the DMPUSHPULL command. See also the DMPUSHPULLSUBTRACT system variable.

Extrude Inside
Determines how to modify the parent entity when it intersects with the extruded/revolved entity when the Auto option of the EXTRUDE and REVOLVE command is selected. See also the EXTRUDEINSIDE system variable.

Extrude Outside
Determines how to modify the parent entity when it touches the extruded/revolved entity when the Auto option of the EXTRUDE and REVOLVE command is selected. See also the EXTRUDEOUTSIDE system variable.

Generate Associative Attributes
Specifies whether associative attributes are generated for 3D entities. See also the GENERATEASSOCATTRS system variable.

Generate Associative Drawings
Specifies associativity between the 3D model and generated views (VIEWBASE) and calculated drawings (BIMSECTIONUPDATE). See also the GENERATEASSOCVIEWS system variable.

Intersected Entities
Determines how to modify entities that intersect with the extruded/revolved entity when the Auto option of the EXTRUDE or REVOLVE command is selected. See also the INTERSECTEDENTITIES system variable.

Report Panel Mode
Specifies the mode of report panel. Applicable only for commands which support it. See also the REPORTPANELMODE system variable.

Selection Modes
Specifies the subentities to highlight during entity selection. See also the SELECTIONMODES system variable.

Sheet Set Manager Auto Open
Toggles whether the **Sheet Sets** panel is opened automatically when a drawing from a sheet set is opened.

Structure Tree Config
Names the CST structure tree configuration file to use.

Unite Surfaces
Determines whether adjacent extruded/revolved surfaces are united or not. See also the UNITESURFACES system variable.

Components Config
Names the CCF components configuration file to use.

Menu Group
Allows you to choose from the available menu groups.

Title
Shows the name of the menu displayed on the menu bar.

Diesel
Executes Diesel code when you select the menu.

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.

Note: 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.

Note: 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.

Note: The list of Aliases and commands will be ordered alphabetically according to the commands when you click **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 **Shell Commands** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: 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.

Prompt
Specifies the prompt to display in the Command line, for example: **Enter file to edit:**.

Flags
Determines how the shell command operates:

- **Don't wait**: BricsCAD immediately returns to the command prompt without waiting for the application to finish.
- **Minimized**: the application runs minimized.
- **Hidden**: the application does not appear on the screen.
- **Quotes**: the shell command uses quotations when the command string uses spaces, such as *C:\cad programs\file name.exe*.

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
The following options appear when you right click on an item:

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.

8.83.5 Manage your customizations

Displays the **Manage Customizations** dialog box to confirm and reverse changes made to the user interface.

8.84 CUTCLIP command

Copies entities to the Clipboard, and then erases them from the drawing.

Shape Lite Pro Mechanical BIM

Icon:

Note: The PICTUREEXPORTSCALE system variable sets the resolution of the image when the exported geometry is pasted in bitmap format, such as in a Word document.

8.84.1 Description

Copies selected entities to the Clipboard for pasting into drawings and other documents. It automatically erases the selected entities from the drawing.

8.85 CVHIDE command

Hides all the control vertices frames from the NURBS curves and/or surfaces.

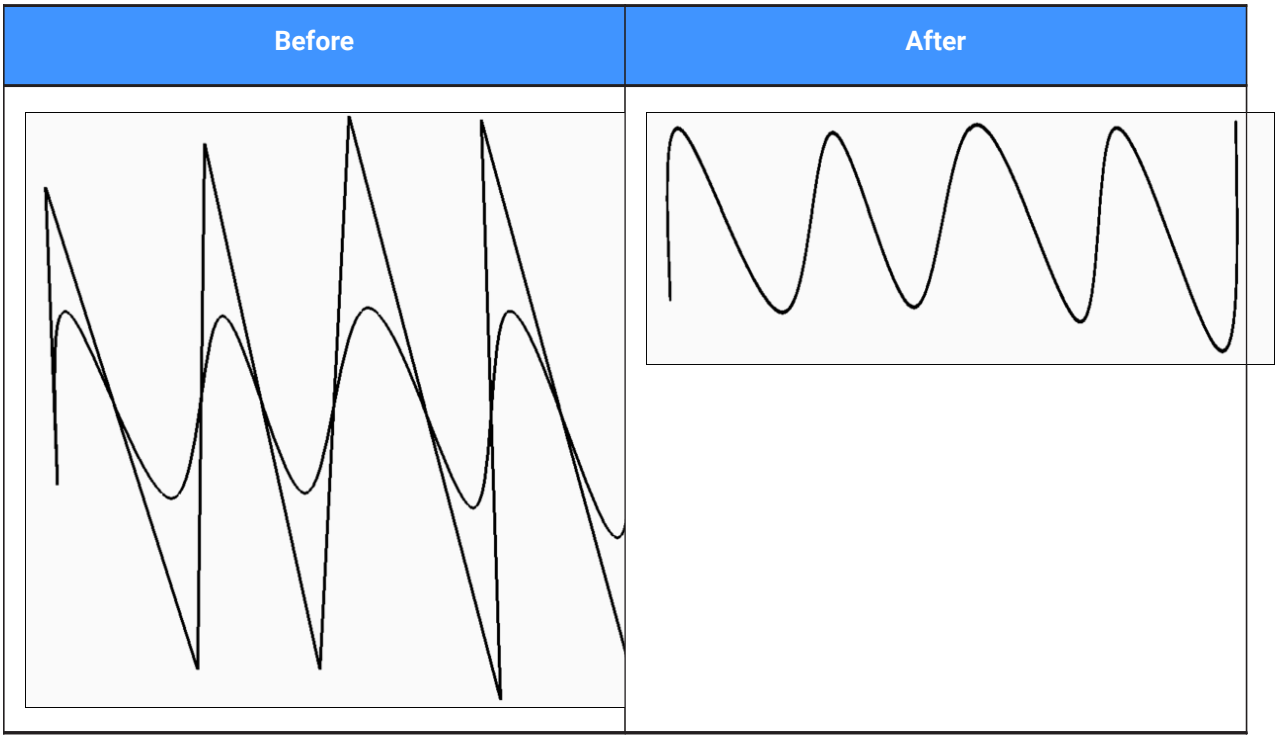
Shape Lite Pro Mechanical BIM

Alias: POINTOFF

8.85.1 Description

Automatically hides all the control vertices frames from the NURBS curves and/or surfaces in the current drawing.

Displays in the Command line the number of NURBS curves and/or surfaces that were processed during the operation.



8.86 CVSHOW command

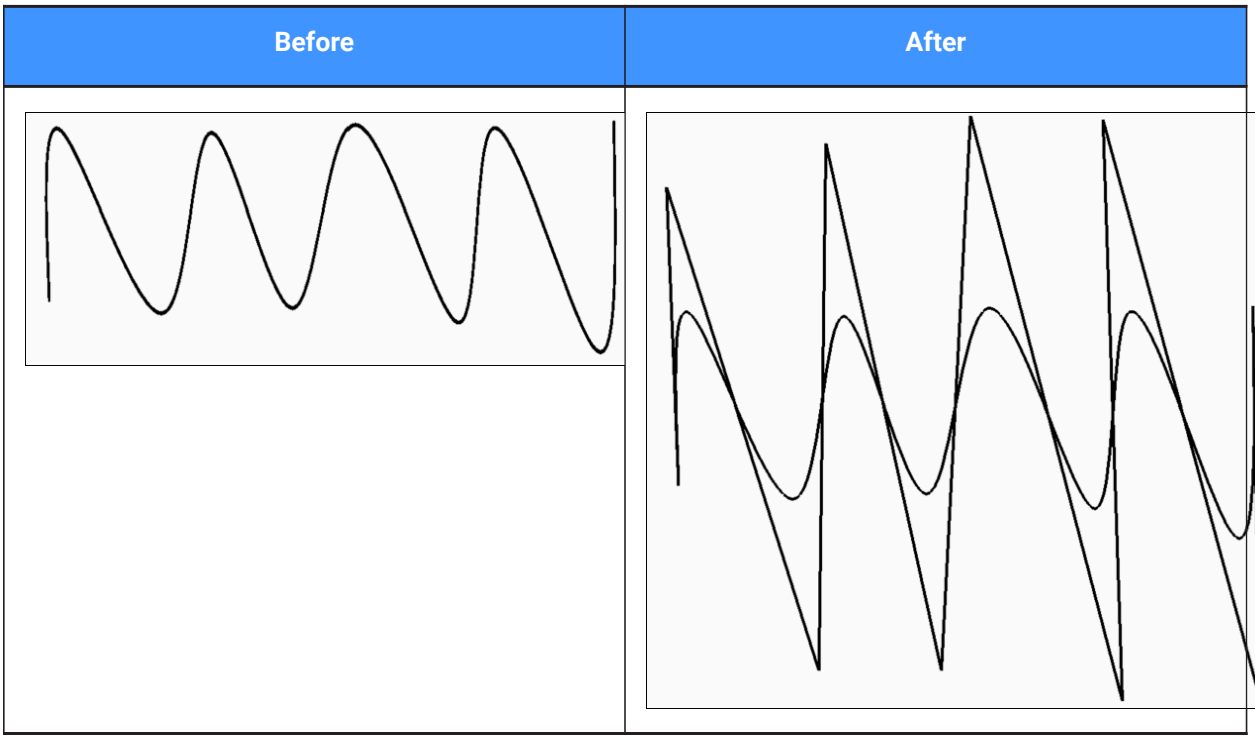
Displays the control vertices frame for the selected NURBS curves and/or surfaces.

Shape Lite Pro Mechanical BIM

Alias: POINTON

8.86.1 Method

Select NURBS curves and/or surfaces to display control vertices frame.



8.87 CYLINDER command

Creates a 3D solid in the shape of a cylinder.

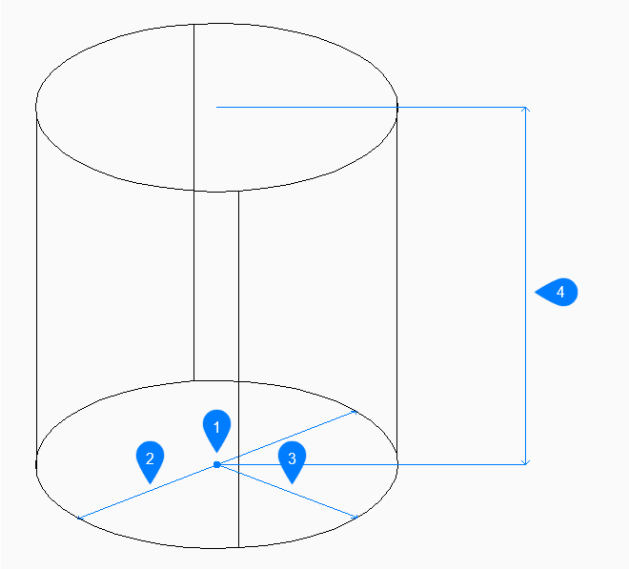
Note: In BricsCAD Lite, which doesn't support 3D solids, the CYLINDER command launches the AI_CYLINDER command.

Shape Lite Pro Mechanical BIM

Alias: CYL

8.87.1 Description

Creates a 3D solid in the shape of a circular or elliptical cylinder. Choose from a combination of options including center, radius, diameter, 3 point, 2 point, tangents, axis endpoints, and height.



1 Center

2 Diameter

3 Radius

4 Height

8.87.2 Method

This command has 5 methods to begin creating a cylinder:

- Center point
- 3 Point
- 2 Point
- Tangent tangent radius
- Elliptical

8.87.3 Options within the command

Select center point
Allows you to begin creating a circular cylinder by specifying the center of the base.

Radius of base of cylinder
Allows you to specify the radius of the cylinder's base.

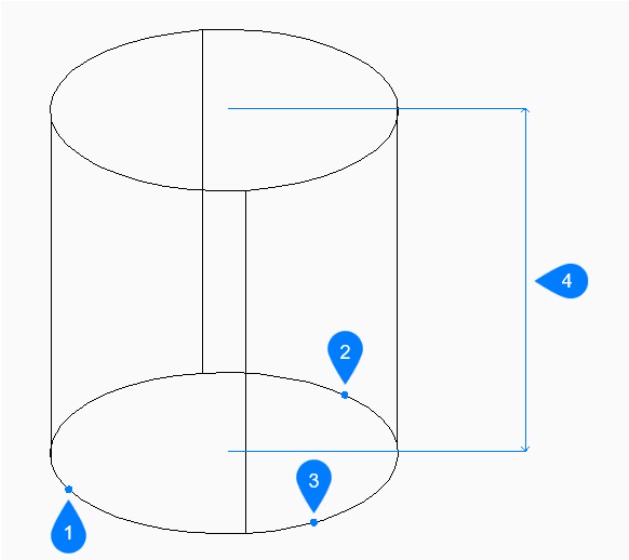
Diameter
Allows you to specify the diameter of the cylinder's base.

3Point
Allows you to begin creating a circular cylinder by specifying three points on the circumference of its base.

First point
Specifies the first point on the circumference.

Second point
Specifies the second point on the circumference.

Third point
Specifies the third point on the circumference.



1 Point 1

2 Point 2

3 Point 3

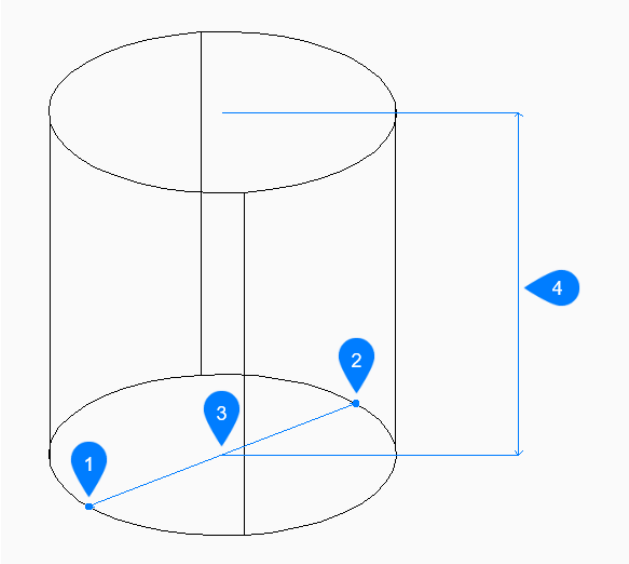
4 Height

2Point
Allows you to begin creating a circular cylinder by specifying two points on the circumference of its base

First end of diameter
Specifies the first point on the diameter.

Second end of diameter
Specifies the second point on the diameter.

Note: The two points define the diameter of the base.



1 Point 1

2 Point 2

3 Diameter

4 Height

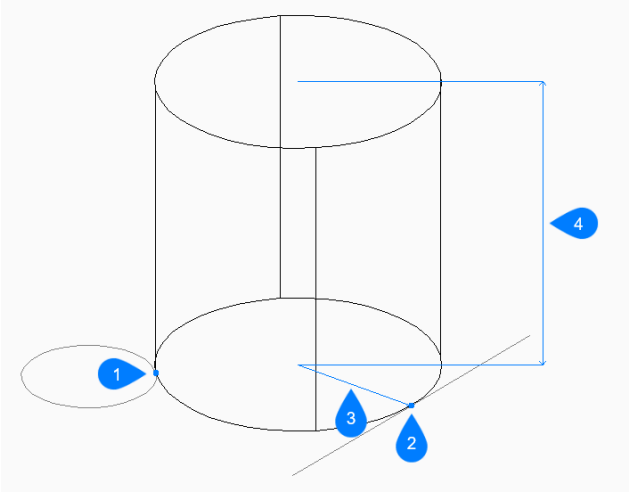
Tangent-tangent-radius
Allows you to begin creating a cylinder by selecting tangent points on the first and second entity, and the radius of the circle.

Specify point on object for first tangent
Allows you to select a tangent point on the first entity.

Specify point on object for second tangent
Allow you to select a tangent point on the second entity.

Radius of circle
Allows you to specify a valid radius for the base.

Note: If you specify a radius that is not possible with the selected tangents, you are prompted to specify the tangents and radius again.



1 Tangent point 1

2 Tangent point 2

3 Radius

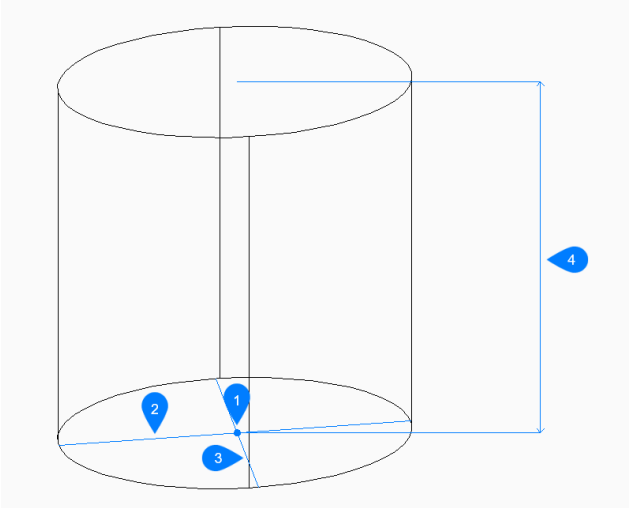
4 Height

Elliptical
Allows you to begin creating an elliptical cylinder by specifying three ends of the ellipse axis.

Set first end of ellipse
Specifies the first end of ellipse axis.

Second end of ellipse
Specifies the second end of the ellipse axis.

Other end of ellipse
Specifies the radius of the other ellipse axis.



1 Center

2 First axis

3 Second axis

4 Height

Specify height
Allows you to specify the height of the cylinder.

2Point
Specifies the height of the cylinder as the distance between any two points.

Axis endpoint
Specifies the axis endpoint to define the height and orientation of the cylinder in 3D space. The center of the base is used as the other axis endpoint.

9. D

9.1 DATAEXTRACTION command

Opens the **Wizard page** dialog box.

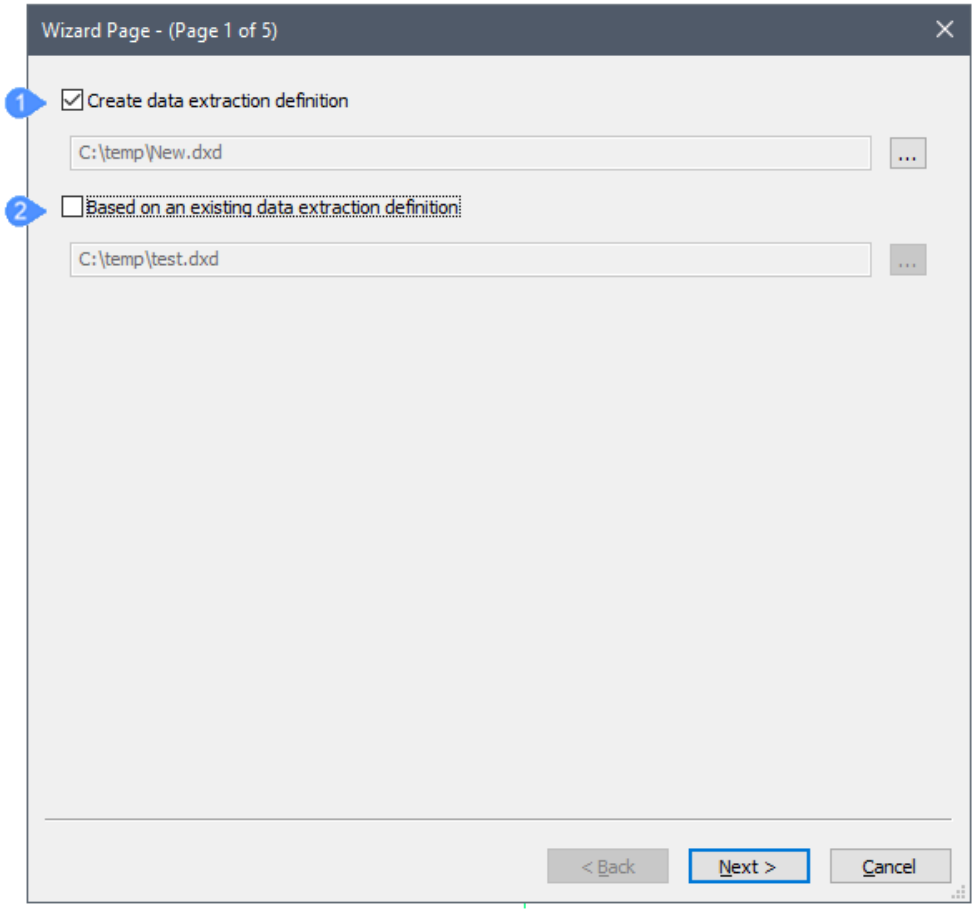
Shape Lite Pro Mechanical BIM

Icon:

9.1.1 Description

Opens the **Wizard Page** dialog box to export entity properties, block attributes, and drawing information to a table or external data file.

9.1.2 Wizard Page 1 Dialog Box



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 **Browse**.

3 Select a folder in the **Save Data Extraction** file dialog box.

4 Enter a name in the **File name** field.

5 Click **Save**.

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 **Browse**.

3 In the **Open Data Extraction Definition** file dialog box, select a .dxd file.

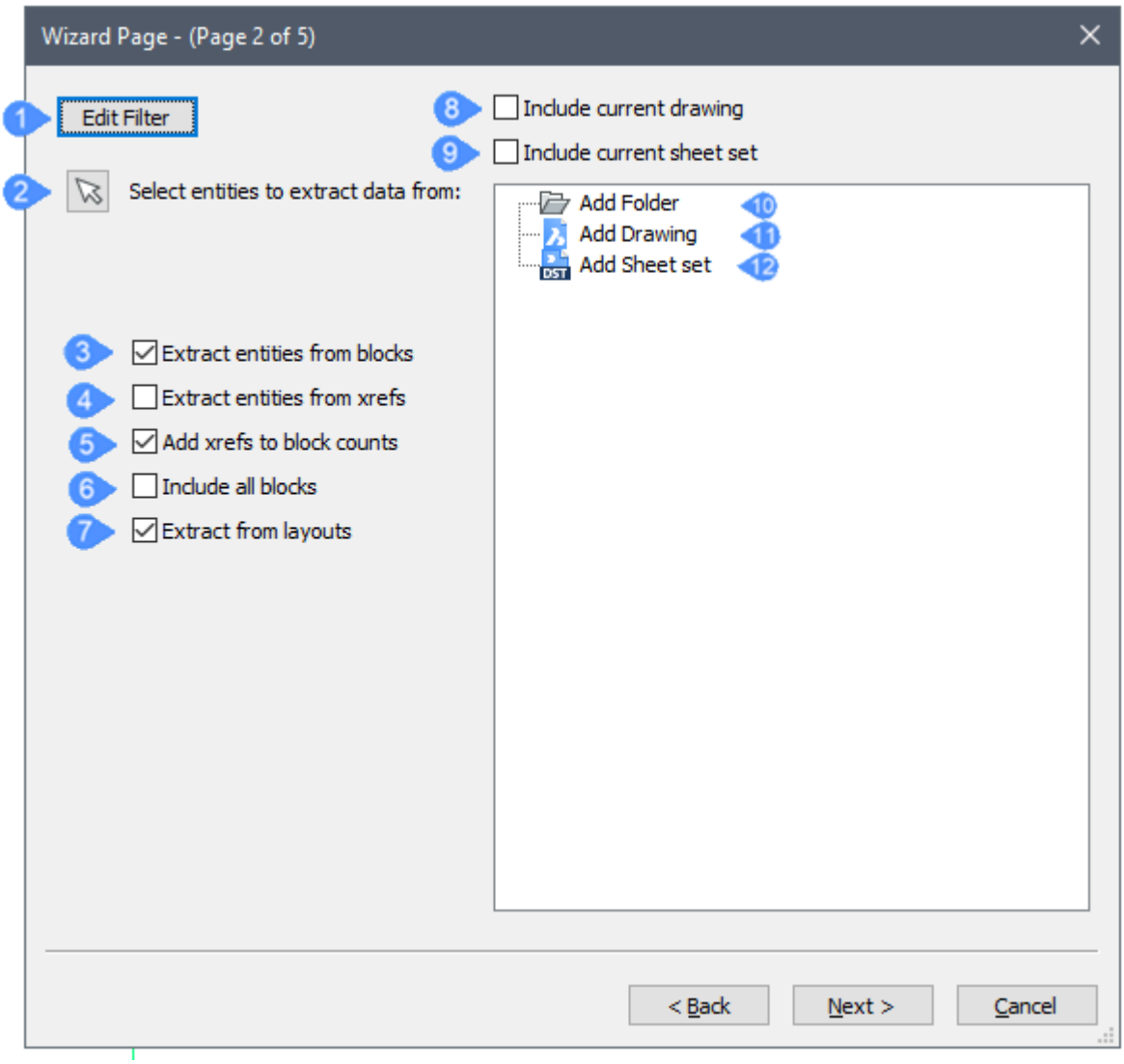
4 Click **Open**.

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.

9.1.3 Wizard Page 2 Dialog Box



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 you 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 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.

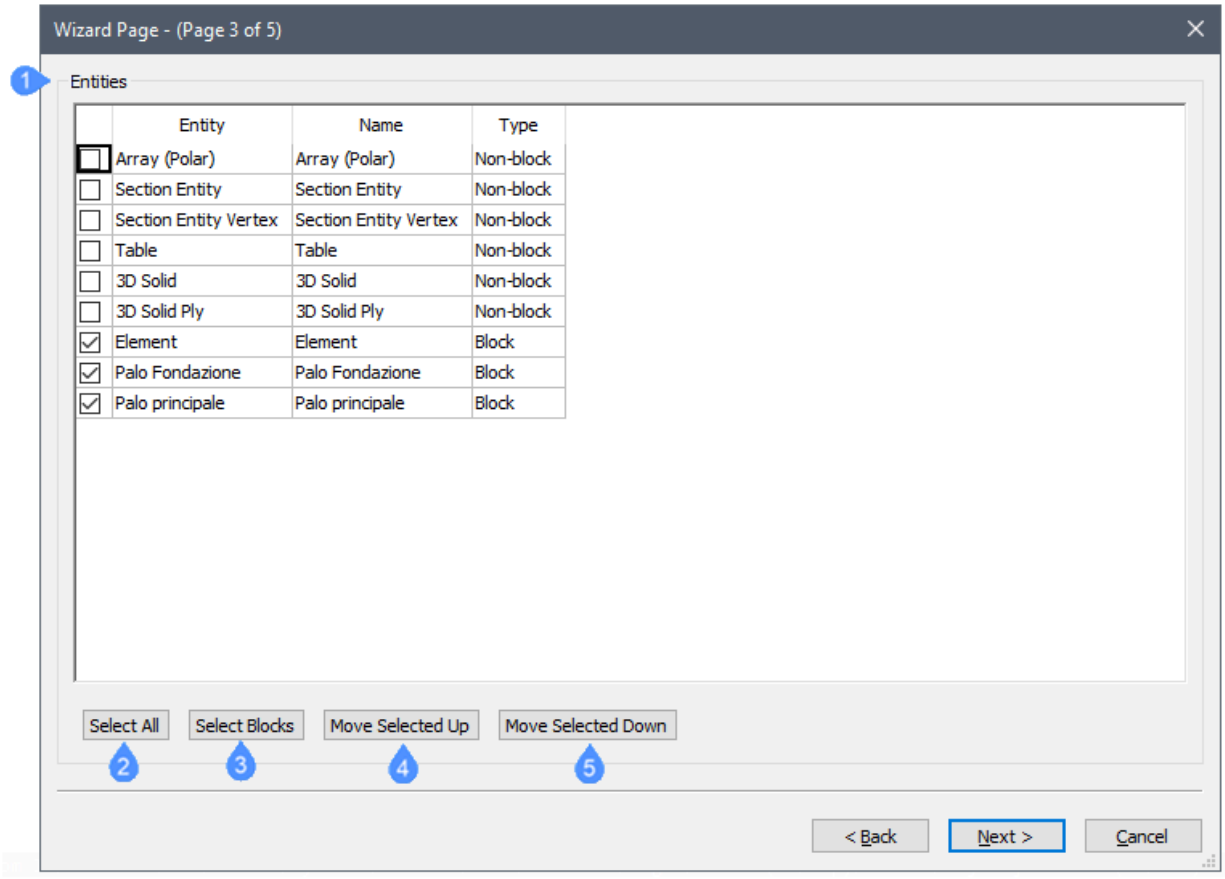
Add Folder
Adds a folder with the drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **Folder**: click **Browse**, 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.

9.1.4 Wizard Page 3 Dialog Box



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, then hold down the **Shift** key while clicking additional items.
- To select multiple items, hold the **Ctrl** key while selecting multiple entities.
- To deselect items, click them again while holding 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.

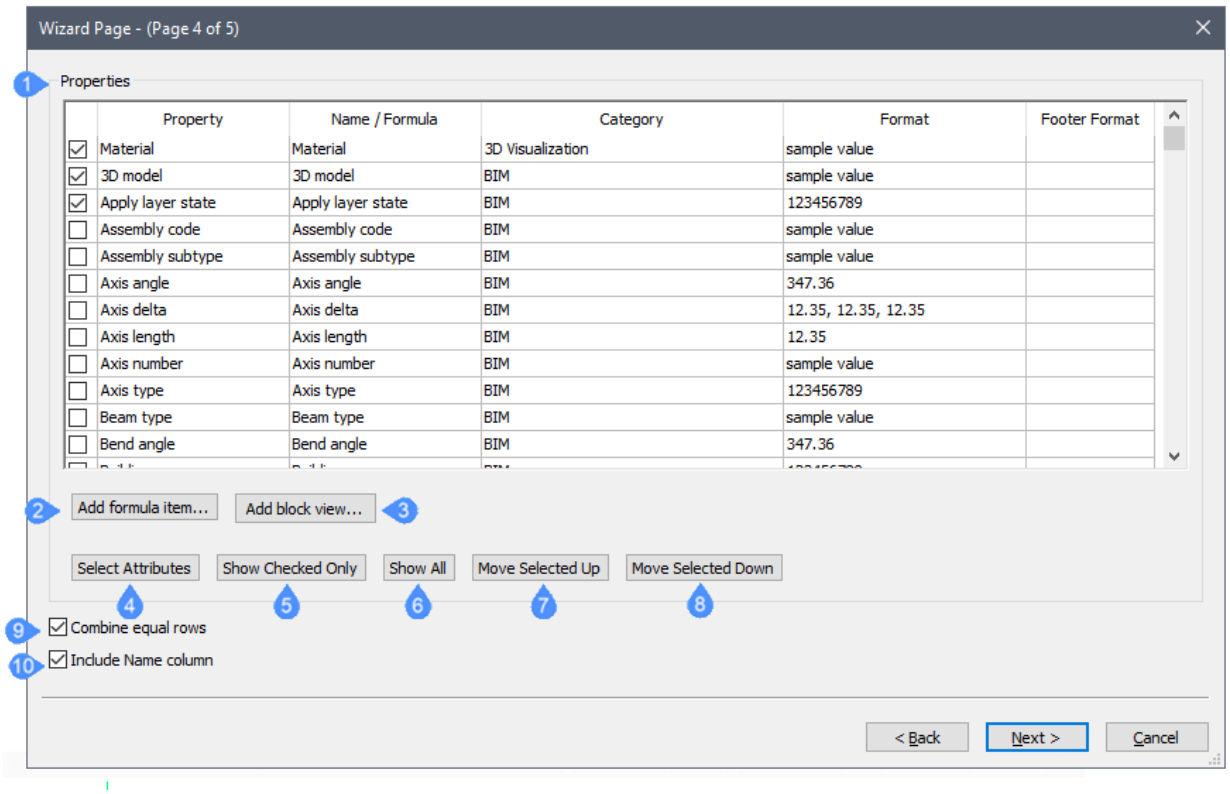
Select Blocks
Selects only blocks in the list, because they usually contain attributes.

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.

9.1.5 Wizard Page 4 Dialog Box



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 box.
- **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 the ones that are not selected.

Show All
Shows all items.

Move Selected Up
Move the selected item(s) up the list.

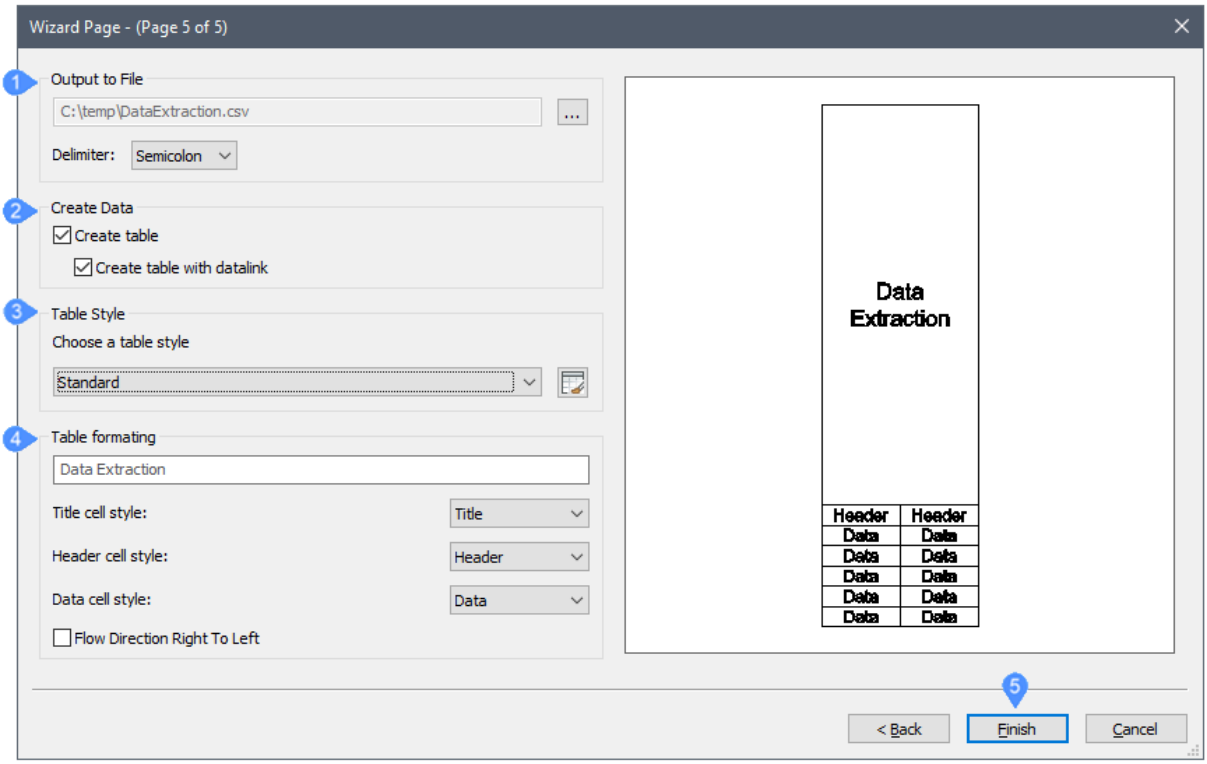
Move Selected Down
Move the selected item(s) down the list.

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.

9.1.6 Wizard Page 5 Dialog Box



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 **Browse**, enter a name for the data file, 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

  Note: 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).

9.2 -DATAEXTRACTION command

Inserts a data extraction table.

Shape Lite Pro Mechanical BIM

9.2.1 Method

Select the data extraction (.dxd) file from the **Generic Open File** dialog box and the insertion point for the table.

9.3 DATALINK command

Opens the **Datalink Manager** dialog box.

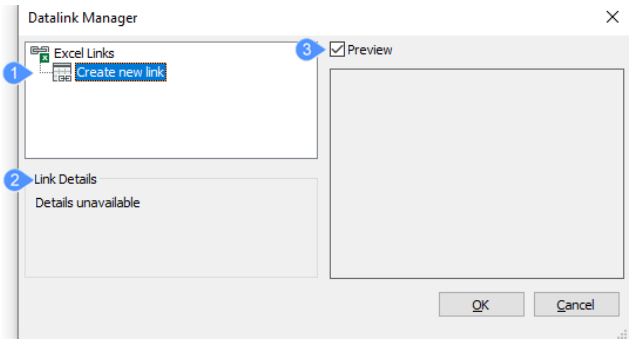
Shape Lite Pro Mechanical BIM

Icon: 

9.3.1 Description

Opens the **Datalink Manager** dialog box to create and manage datalinks in the current drawing.

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

9.3.2 Create new link

Opens the **DataLink Edit** dialog box. See the related article **DataLink Edit dialog box**.

9.3.3 Link details

Displays link details.

9.3.4 Preview

Display a preview of the datalink.

9.4 DATALINKUPDATE command

Synchronizes the linked data in tables in the current drawing with data in the linked source file.

Shape Lite Pro Mechanical BIM

Icon:

9.4.1 Method

There are two methods to use DATALINKUPDATE command:

- Update the data link existing in a table in the drawing.
- Update the linked data in an external file.

9.4.2 Options within the command

Update data link
Updates the linked data in a table in the drawing with data that has been changed in the external source file.

Write data link
Updates the linked data in an external file with data that has been changed in a table in the drawing.

Select objects
Prompts you to select table entities.

Update all
Synchronizes all linked data in all tables in the drawing.

9.5 DATASMITHCONNECT command

Creates a connection to which Twinmotion and/or Unreal Engine can connect.

Shape Lite Pro Mechanical BIM

Icon:

9.5.1 Description

Creates a datasmith connection, which can be picked up by Twinmotion or Unreal Engine to establish a Direct Link.

If a connection was already established for another drawing, it will be removed and replaced by the new connection.

Note:

- After establishing a connection to Twinmotion with Direct Link, the model is automatically synchronized the first time.
- Use the DATASMITHSYNC command to synchronize the latest changes of the current drawing to the Direct Link.

9.6 DATASMITHEXPORT command

Exports the current drawing to a udatasmith file.

Shape Lite Pro Mechanical BIM

Icon:

9.6.1 Description

Opens the **Export drawing to** dialog box to save data from the current drawing to a Unreal Engine file type (*.udatasmith) file, which can be used in Twinmotion and/or Unreal Engine.

Note: When exporting to Twinmotion/Datasmith, a 3D visual style should be used, not the 2dWireframe.

9.7 DATASMITHSYNC command

Sends changes to Twinmotion and/or Unreal Engine since the last call to the connection.

Shape Lite Pro Mechanical BIM

Icon:

9.7.1 Description

If a datasmith connection for a Direct Link with Twinmotion and/or Unreal Engine exists, it synchronizes it to reflect the latest changes in the current drawing.

Note: When there is no connection yet, use the DATASMITHCONNECT command first.

9.8 DATE command (Express Tools)

Displays the date and time in the Command line.

Shape Lite Pro Mechanical BIM

9.8.1 Method

The date and time are automatically displayed in the Command line.

```
Thu Sep 15, 2022  10:20:37
```

9.9 DBLIST command

Lists information about all entities in the drawing (short for "database listing").

Shape Lite Pro Mechanical BIM

9.9.1 Description

Displays information about every entity from the drawing.

The information can refer to handle, current space, layer, color, etc.

Note: Press F2 to see the data listed in the Prompt History window. For drawings with many entities, the list can take a long time to finish. Press Esc to stop the command.

9.10 DCALIGNED command

Constrains the distance between two entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.10.1 Description

Constrains the shortest distance between a point and an entity. Constrains the length of a line, a polyline segment, or an arc. Constrains the distance between two lines, making them parallel. Only the distance is constrained (locked); entities can still be moved (in tandem), rotated, and so on. Aligned constraints look like aligned dimensions.

9.10.2 Method

There are three methods to begin creating an aligned constraint:

- Entity
- Point & line
- 2Lines

9.10.3 Options within the command

Entity
Constraints the length of a line, a polyline segment or the chord of an arc.

Note: The entity can still be modified (moved, rotated), even if the length is constrained.

Point & line
Constraints the perpendicular distance between a point and a line or polyline segment.

Note: A valid constraint point is usually the same geometric location as entity snaps, such as the ends and middles of lines, centers of circles and arcs, and so on.

Note: The constrained distance can be edited directly or with the Properties panel.

2Lines
Constrains the distance between two straight linear segments. If the entities are not parallel, then the entity picked second is rotated around its midpoint to be parallel to the first entity.

Note: The constrained distance can be edited directly or with the **Properties** panel.

9.11 DCANGULAR command

Constrains angles.

Shape Lite Pro Mechanical BIM

Icon:

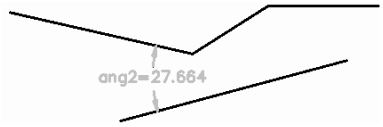
9.11.1 Description

Constrains the angle between two lines or straight polyline segments; also constrains the angles of arcs, polyline arcs, and any three constraint points.

9.11.2 Methods

There are three methods to begin constraining angles:

Between two straight segments
Constrains the angle between two segments.

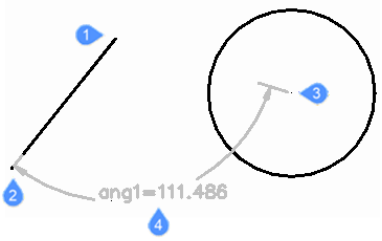


Of arcs or polyline arcs
Constrains the angle of an arc between its endpoints.



Between three valid constraint points

Valid constraint points are at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.



1 Angle vertex

2 First angle constraint point

3 Second angle constraint point

4 Dimension location

9.12 DCCONVERT command

Turns associative dimensions into constraints.

Shape Lite Pro Mechanical BIM

Icon:

9.12.1 Description

Converts dimensions to the appropriate dimensional constraints, such as linear dimensions into linear constraints, or diameter dimensions into diameter constraints. Dimensional constraints are colored gray.



9.13 DCDIAMETER command

Constrains diameters.

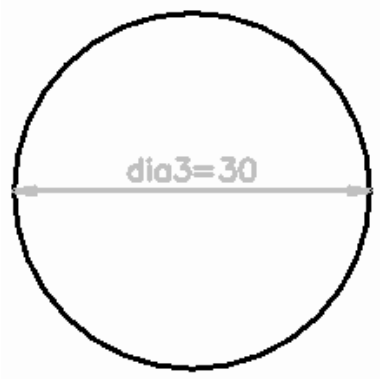
Shape Lite Pro Mechanical BIM

Icon:

9.13.1 Description

Constrains the diameters of circles, arcs, and polyline arc segments.

Although the diameter of the entity is fixed (constrained), the entity can be moved, rotated, and so on.



9.14 DCDISPLAY command

Toggles dimensional constraint visibility.

Shape Lite Pro Mechanical BIM

Icon:

9.14.1 Description

Toggles the display of dimensional constraints (attached to selected entities) between visible and hidden. All dimensional constraints are initially hidden when a drawing containing them is opened. Many constraints in a drawing can clutter it, and so it is useful to hide them.

9.15 DCHORIZONTAL command

Constrains entities horizontally.

Shape Lite Pro Mechanical BIM

Icon:

9.15.1 Description

Horizontally constrains the distance between two points, or the length of a single entity.

Note: "Horizontal" means the constraint lies in the X direction of the current coordinate system.

9.15.2 Method

There are two methods to begin constraining horizontally:

- By distance: constrain the distance between two entities horizontally.
- ***Note:*** Valid constraint points are located at same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.
- By entity: constrain an entity horizontally.

  Note: Constrains the following entities horizontally: Line, Arc, Polyline segment, Polyline arc segment.

9.15.3 Options within the command

Entity
Constrains the horizontal distance between the endpoints of an entity.

9.16 DCLINEAR command

Constrains linearly.

Shape Lite Pro Mechanical BIM

Icon: 

9.16.1 Description

Constrains the distance between two points, or the length of a single entity, to the horizontal or vertical, depending how the cursor is moved, as in the DIMLINEAR command.

Note: "Horizontal" means the constraint lies in the X direction of the current coordinate system, and "vertical" in the Y direction.

9.16.2 Method

There are two methods to begin constraining linearly:

- By distance: constrain the distance between two entities vertically or horizontally.

  Note: Valid constraint points are found at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.
- By entity: constrain the length of an entity between its endpoints, vertically or horizontally.

  Note: Constrains one of the following entity types horizontally or vertically: Line, Arc, Polyline segment, Polyline arc segment.

The program applies a vertical or horizontal constraint, depending on how the cursor is moved during the command.

9.16.3 Options within the command

Entity
Constrains the horizontal or vertical distance between the endpoints of an entity.

9.17 DCRADIUS command

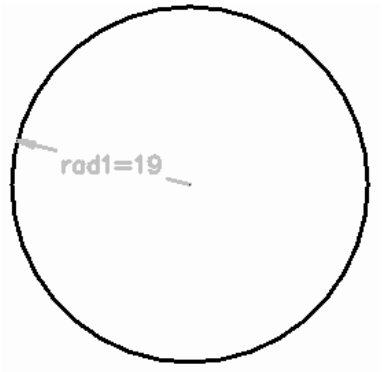
Constrains radii.

Shape Lite Pro Mechanical BIM

Icon: 

9.17.1 Description

Constrains the radius of circles, arcs, or polyline arc segments. Although the radius of the entity is fixed (constrained), the entity can be moved, rotated, and so on.



9.18 DCVERTICAL command

Constrains entities vertically.

Shape Lite Pro Mechanical BIM

Icon:

9.18.1 Description

Vertically constrains the distance between two points, or the length of an entity.

Note: "Vertical" means the constraint lies in the Y direction of the current coordinate system.

9.18.2 Method

There are two methods to begin constraining vertically:

- By distance: constrain the distance between two entities vertically.

  Note: Valid constraint points are found at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.

- By entity: constrain an entity vertically.

  Note: Constrains any of the following entities vertically: Line, Arc, Polyline segment, Polyline arc segment.

To constrain the distance between two entities vertically, select a valid constraint point on each entity and enter the constraint distance. The value controls the distance between the two entities.

9.18.3 Options within the command

Entity

Constrains the vertical distance between the endpoints of an entity.

9.19 DDATTE command

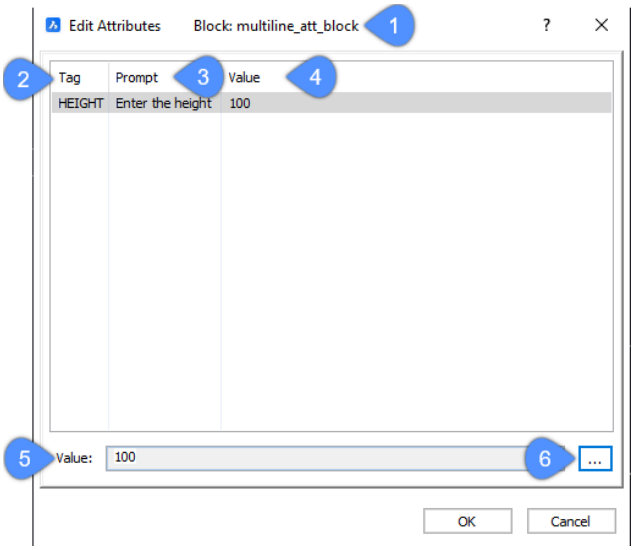
Edits the values of attributes through a dialog box. This command is superseded by the BATTMAN command.

Shape Lite Pro Mechanical BIM

9.19.1 Description

Opens the **Edit Attributes** dialog.

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

9.19.2 Block name

Displays the name of the selected block.

9.19.3 Tag

Specifies the name of the attribute.

Note: This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.

9.19.4 Prompt

Specifies the user prompt.

Note: 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.

9.19.5 Value

Displays the value that belongs to a certain attribute.

9.19.6 … Text Formatting

Changes the value of the attribute text.

Note: 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.

9.20 DDEDIT command

Edits single-line text, multi-line text (mtext), dimension text, attribute definition and leader text.

Shape Lite Pro Mechanical BIM

Alias: ED

9.20.1 Description

Allows editing single-line texts, multi-line text (mtext), dimension text, attribute definitions, and leader text.

Note: The DDEDIT command is also launched when you double-click one of the mentioned entities.

If the selected entity is represented by multi-line text, dimension text or leader text, the **Text formatting** window opens.

If the selected entity is represented by an attribute definition, the **Edit Attribute Definition** dialog box opens.

If the selected entity is represented by single-line text, the in-place editor is displayed (this editor has no User Interface).

Note: Change the value of TEXTED system variable to change the editing method of a single-line text.

9.21 DDEMODES command

Sets default values for creating entities.

Shape Lite Pro Mechanical BIM

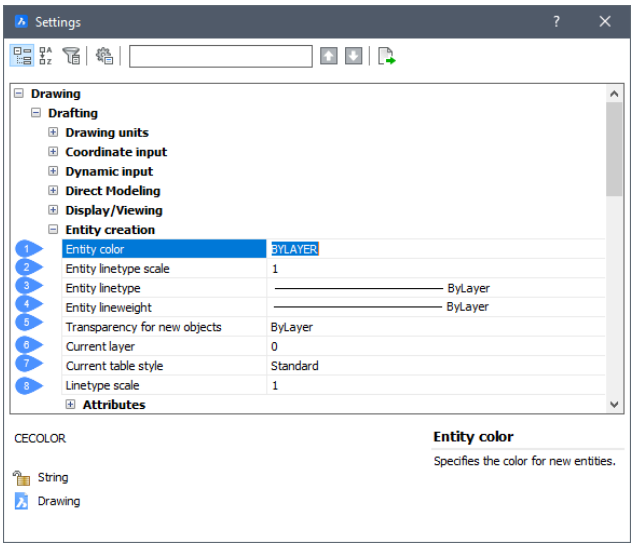
9.21.1 Description

Displays the Entity Creation section of the **Settings** dialog box.

Note: This command is superseded by the SETTINGS command.

9.21.2 Method

Displays **Settings** dialog box at the **Entity creation** section:



1 Entity color

2 Entity linetype scale

3 Entity linetype

4 Entity lineweight

5 Transparency

6 Current layer

7 Current table style

8 Linetype scale

Entity color
Specifies the default color of new entities; initial value is ByLayer, which means that entity colors are governed by layer properties.

Entity linetype scale
Specifies the default scale factor for linetypes; initial value is 1.0000.

Entity linetype
Specifies the default linetype for new entities; initial value is ByLayer, which means that entity linetypes are governed by layer properties.

Entity Lineweight
Specifies the default lineweights for new entities; initial value is ByLayer, which means that entity lineweights are governed by layer properties.

Transparency
Specifies and control the transparency of entities.

Current layer
Specifies the default layer for new entities; initial value is layer 0.

Current table style
Specifies the initial value of the table style name for new tables.

Linetype scale
Specifies the global linetype scale factor; this factor affects entity linetype scale factors.

9.22 DDFILTER command

Creates a selection set of the entities.

Shape Lite Pro Mechanical BIM

Note: This command is superseded by the SELECT command.

9.22.1 Description

Creates a selection set that include the selected entities, so you may access it by entering "P" (previous) the next time a command prompts you "Select entities."

9.23 DDGRIPS command

Opens the **Settings** dialog box with the **Grips** category expanded.

Shape Lite Pro Mechanical BIM

Alias: GR

9.23.1 Description

Opens the **Settings** dialog box with the **Grips** category expanded to view and modify relevant system variables.

9.24 DDPTYPE command

Opens the **Settings** dialog box with the **Points** category expanded.

Shape Lite Pro Mechanical BIM

9.24.1 Description

Opens the **Settings** dialog with the **Points** category expanded to view and modify relevant system variables.

9.25 DDSELECT command

Opens the **Settings** dialog box with the **Entity selection** category expanded.

Shape Lite Pro Mechanical BIM

Alias: SE

9.25.1 Description

Opens the **Settings** dialog box with the **Entity selection** category expanded to view and modify relevant system variables.

9.26 DDSETVAR command

Opens the **Settings** dialog box.

Shape Lite Pro Mechanical BIM

9.26.1 Description

Opens the **Settings** dialog box to view and modify system variables. Most, but not all, system variables are available in the **Settings** dialog box. You can modify all system variables using the SETVAR command.

9.27 DDSTRACK command

Opens the **Settings** dialog box with the **Snap tracking** category expanded.

Shape Lite Pro Mechanical BIM

9.27.1 Description

Opens the **Settings** dialog box with the **Snap tracking** category expanded to view and modify relevant system variables.

9.28 DDVPOINT command

Opens the **Set Viewpoint** dialog box.

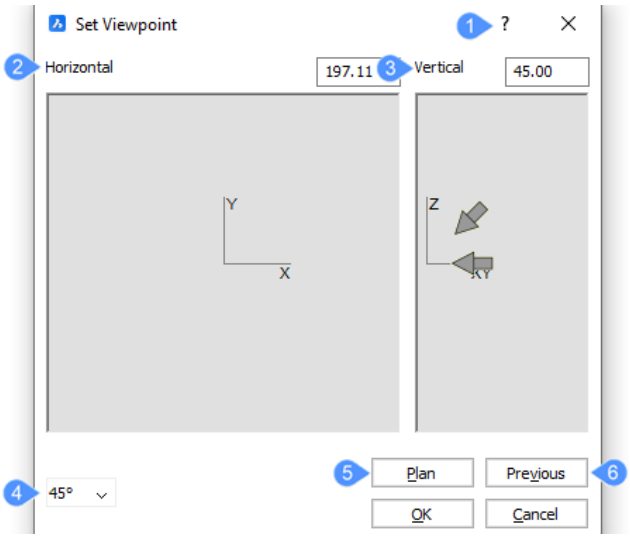
Shape Lite Pro Mechanical BIM

Alias: SETVPOINT, VIEWCTL, VP

9.28.1 Description

Opens the **Set Viewpoint** dialog box to specify a 3D viewpoint for the current viewport.

Note: 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

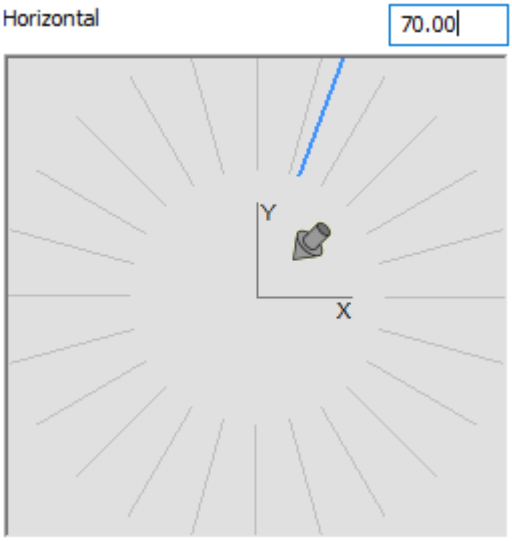
6 Previous

9.28.2 Command reference

Opens the Bricsys Help article about the DDVPOINT command.

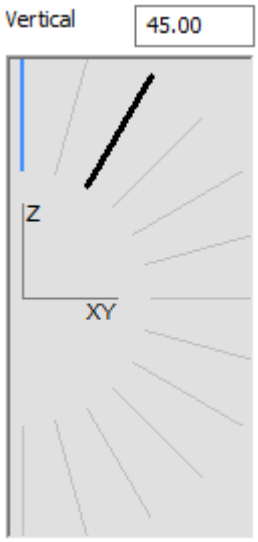
9.28.3 Horizontal

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.

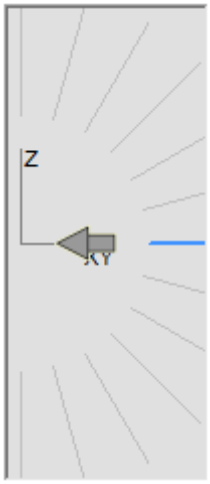


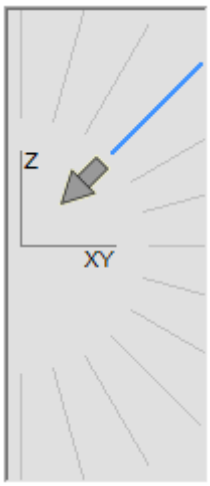
9.28.4 Vertical

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

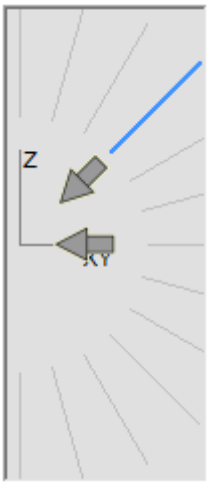


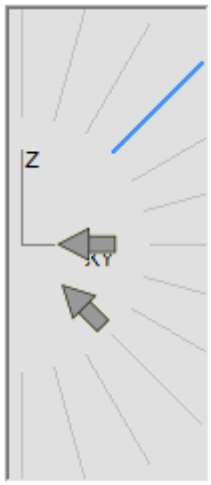
Click in the inner half circle. (1) View direction is horizontal. (2) View direction is 45° down (not available if PresetAngle = 45°).

(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) 

9.28.5 Angle Preset

Sets markers at 5, 15, or 45 degrees; default = 45.

9.28.6 Plan

Reverts the 3D viewpoint to 2D plan view.

9.28.7 Previous

Reverts to the previous viewpoint.

9.29 DEACTIVATESTORY command

Deactivates an active story.

Shape Lite Pro Mechanical BIM

9.29.1 Description

Automatically deactivates an active story without any view modification.

9.30 DEFAULTSCALELIST command

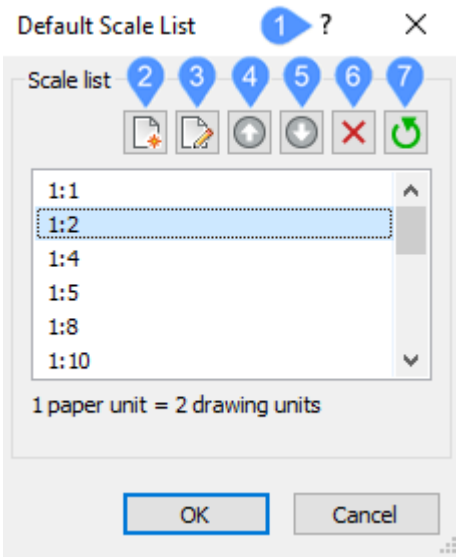
Opens the **Default Scale List** dialog box.

Shape Lite Pro Mechanical BIM

9.30.1 Description

Opens the **Default Scale List** dialog box to view and manage the default scale list. The default scale list is stored in the registry. When you reset the current drawing's scale list, it updates to match the default scale list.

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

9.30.2 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

9.30.3 Add new scale

Allows you to add a new scale factor to the list. Opens the **Add Scale** dialog box.

9.30.4 Edit scale

Allows you to edit the factors of an existing scale. Opens the **Edit Scale** dialog box.

9.30.5 Move up

Moves the selected scale factor up the list.

9.30.6 Move down

Moves the selected scale factor down the list.

9.30.7 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.

9.30.8 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.

9.31 DELAY command

Delays execution of commands.

Shape Lite Pro Mechanical BIM

9.31.1 Description

This command is used to delay the execution of the next command.

Note: Meant to use with scripts.

9.31.2 Options within the command

Milliseconds to delay

Specifies the amount of time BricsCAD waits before continuing to the next command in the script.

Note: Enter a number between 0 and 2,147,483,627(approximately 24 days).

9.32 DELCONSTRAINT command

Removes dimensional and geometrical constraints from selected entities.

Shape Lite Pro Mechanical BIM

9.32.1 Method

Select one or more entities or type ALL to select all entities in the drawing to delete all their dimensional and geometric constraints.

Note: To remove individual constraints one at a time:

- Dimensional constraints: select the dimensional constraint, and then press ERASE command.
- Geometric constraints: click the tiny x on the constraint bar: 

9.33 DELEDATA command

Deletes extended entity data for specific applications from the selected entities (short for "delete entity data").

Shape Lite Pro Mechanical BIM

9.33.1 Description

Specifies the name of the application to which the entity data belongs and selects one or more entities from which Entity Data will be deleted.

9.33.2 Options

? to list application names

Lists names of applications loaded in the current drawing.

9.34 DELETESKETCHFEATURE command

Deletes the sketch features and their corresponding solids.

Shape Lite Pro Mechanical BIM

9.34.1 Description

Allows you to delete sketch features and all their dependencies.

Note: The EXTRUDE, LOFT, SWEEP, and REVOLVE commands and their children features SUBTRACT, and UNITE, create Features when the CREATESKETCHFEATURE system variable is ON. They are available in the **Mechanical Browser** panel (for example, Extrude_1, Loft_1).

9.34.2 Method

Specify a comma separated list of Sketch Based Features (for example, Extrude_1, Loft_1) to be deleted. Otherwise, the command will delete all existing Sketch Based Features, if there is more than one.

Note: Deletes all corresponding sketch features and their corresponding solids in the current drawing.

9.35 DEPARAMETRIZE command

Deparametrizes the selected entities.

Shape Lite Pro Mechanical BIM

9.35.1 Description

Deparametrizes the selected entity. This command can also be used with parametrized 2D entities.

Note: Make sure the parametrized entities are converted to blocks before running the command.

9.35.2 Options within the command

Once the command is initiated, options regarding the selection of entities to deparametrize are available.

Select block references to make static
Select block references in the drawing to deparametrize. Only the selected entities will be made static.

Entire drawing
Deparametrizes all entities in the current drawing.

Note: To verify whether the block reference is made static or not, check the properties of the selected block in the **Properties** panel.

9.36 DESIGNTABLE command

Imports CSV files that contain sets of parameter values or else creates them from scratch.

Shape Lite Pro Mechanical BIM

Icons:

9.36.1 Description

Imports CSV files that contain sets of parameter values or creates them from scratch. Parameters are then added to the **Mechanical Browser** panel and to parametric component properties in **Properties** panel.

Note: This command can be entered transparently during the commands (‘designtable).

9.36.2 Options within the command

From file
Creates a design table by importing a CSV file (comma-separated values file). Make sure the list separator character on your system is the same as the one used in the CSV file.
Displays the **Select Design Table File** dialog box from where you will select the CSV file and open it.

Note: All parameters defined in the CSV file are created automatically.

Note: Enter the MECHANICALBROWSEROPEN command to view the constraints.

Empty
Creates an empty design table from scratch.

Note: Enter the names of the parameters to include in the design table or choose to add all parameters that have a constant value. The constant parameters do not depend on another parameter.

9.37 -DESIGNTABLEEDIT command

Edits design tables.

Shape Lite Pro Mechanical BIM

Note: This command can be entered transparently during the commands (‘designtable).

9.37.1 Options within the command

eXport
Exports the content of a design table entity in CSV format file.

Replace
Replaces the content of a design table with data from a CSV format file.

Delete
Deletes a design table.

Configurations
Creates, removes, or views configurations.

Save current
Creates a new configuration as a copy of an existing one.

Remove
Removes a configuration.

? to list
Opens the Prompt History window. Displays a list of available configurations, the current one in use and the total number of configurations.

Parameters
Manages parameters.

Unlink
Unlinks parameters from the design table.

Link
Links parameters to the design table.

Note: **(*) to link all**: restores all initially existent but removed parameter links.

Clear expressions
Clears expressions (values and formulae) from parameters.

? to list columns
Opens the **Prompt History** window, then displays two lists and the total number of columns.
The first list contains the columns linked to the parameters.
The second list contains the columns not linked to the parameters.

Apply configuration
Creates a new row using the current values of variables.

? to list
Lists design tables attached to the current drawing.

9.38 DETAILSPANELCLOSE command

Closes the **Details** panel.

Shape Lite Pro Mechanical BIM

9.38.1 Description

Closes the **Details** panel to hide it from the current workspace. If the **Details** panel is stacked when you close it, the Details tab or icon is removed from the stack.

9.39 DETAILSPANELOPEN command

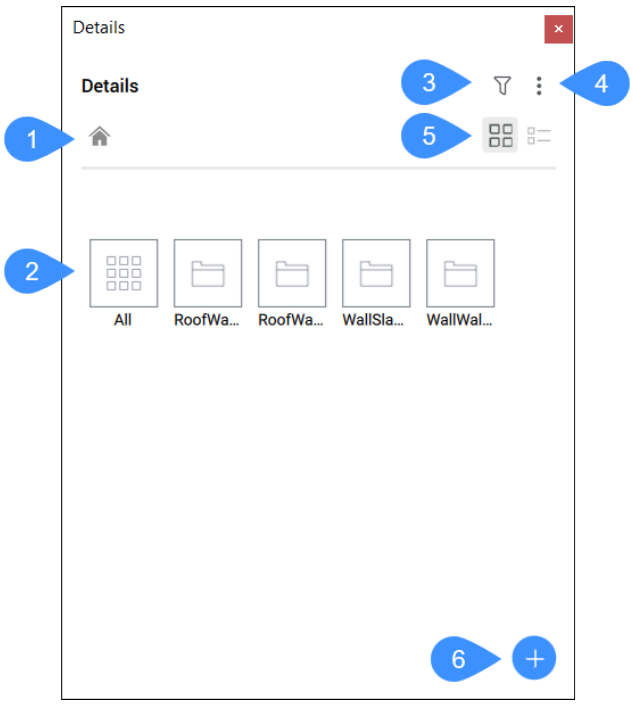
Opens the **Details** panel.

Shape Lite Pro Mechanical BIM

9.39.1 Description

Opens the **Details** panel to display it in the current workspace. The **Details** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Details** panel can be either floating, docked or stacked.

The **Details** panel allows you to view details, add tags to details, propagate and adjust details.



1 Home

2 Details library

3 Filter

4 Menu

5 List view / Grid view

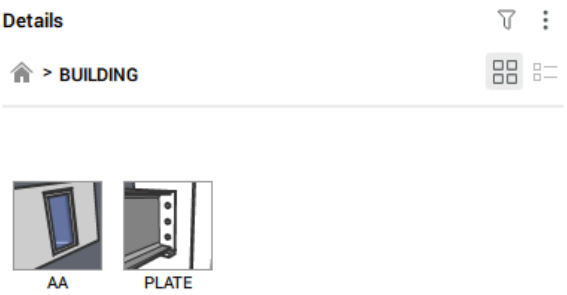
6 Create detail

9.39.2 Home

Click this button to go back to the details library overview.

9.39.3 Details library

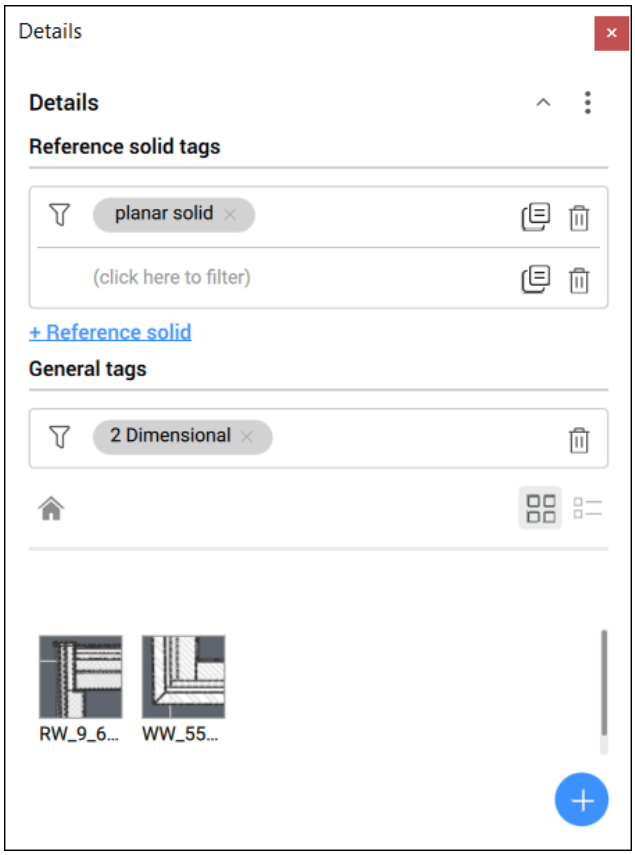
You can click on the category of the detail you want to view. The details assigned to that category will be displayed.



When you click on a detail, the **View Detail** dialog box will appear (see **View Detail dialog box** article).

9.39.4 Filter

Click the filter icon to open the filter options. Here you can search the library for specific details using tags.



Reference solid tags
Filters the details based on Reference solid tags. Click **+ Reference solid** to add a filter and then select a filter from the drop-down list. You can copy or delete filters by clicking on the symbols on the right.

General tags
Filters the details based on General tags. Click on **(click here to filter)** to add filters. Select a tag from the drop-down list. Click the cross next to a filter to delete a filter, or click the delete icon to the left to delete all the added filters at once.

9.39.5 Menu

Generate thumbnails
Loads the new details in your **Library** panel.

Manage libraries
Opens the **Settings** dialog box to change the details directory path.

9.39.6 List view / Grid view

Toggles between the list view and the grid view of the details by clicking the icons.

9.39.7 Create detail

Launches the BIMCREATEDETAIL command.

9.40 DGNEXPORT command

Exports the current drawing to a DGN file format.

Shape Lite Pro Mechanical BIM

9.40.1 Description

Opens the **Export Drawing As** dialog box to export the current drawing to a MicroStation DGN File (*.dgn) format.

9.41 -DGNEXPORT command

Exports the current drawing to MicroStation DGN File (*.dgn) format in the command line.

Shape Lite Pro Mechanical BIM

9.41.1 Method

Specify the full path and file name for the exported DGN file.

Note: Type ~ (tilde) to display the **Export Drawing As** dialog box, which allows you to specify the folder and the name for the DGN file.

9.41.2 Options within the command

DGN
Converts reference to DGN.

DWG
Keeps reference to DGN.

Bind
Binds reference file to the parent DGN.

Detach
Detaches external reference.

9.42 DGNIMPORT command

Imports MicroStation DGN files into the current drawing.

Shape Lite Pro Mechanical BIM

9.42.1 Description

Opens the **Import file** dialog box to select a .dgn file to import into the current drawing.

Note: Run the DGNIMPORTOPTIONS command to open the **Settings** dialog box with the DGNIMPORT category expanded. All options affecting the import of .dgn files can be modified here.

9.43 DGNIMPORTOPTIONS command

Opens the **Settings** dialog box with the **DGNIMPORT** category expanded.

Shape Lite Pro Mechanical BIM

9.43.1 Description

Opens the **Settings** dialog box with the **DGNIMPORT** category expanded to view and modify relevant system variables.

9.44 DIGITALSIGN command

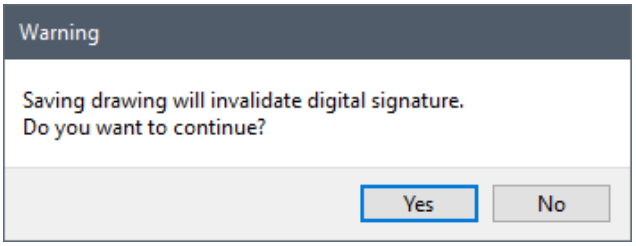
Applies a digital signature (block of encrypted information) to the drawing.

Shape Lite Pro Mechanical BIM

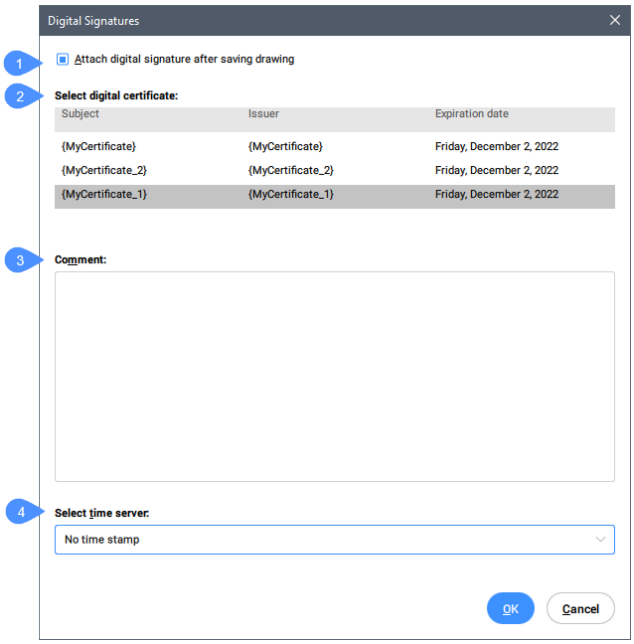
9.44.1 Method

Opens the **Digital Signatures** dialog box where you can attach a digital signature to the drawing in order to validate the origin, authenticity, and unaltered state of the file since the digital signature was applied.

Note:

- The digital signature remains valid after renaming the drawing.
- The SIGWARN system variable controls the display of a dialog with the signature content when a drawing with a digital signature is opened.
- After the DIGITALSIGN command is launched, a digital signature is applied every time when saving the drawing until it is closed. The next time the signed drawing is opened and modified, a warning message appears when saving.
- 

Applies a digital signature to the drawing.



Attach digital signature after saving drawing

Tick the box to attach the digital signature after saving the drawing.

Select digital certificate

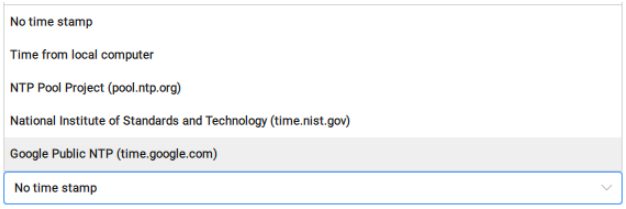
Select the digital certificate that you want to attach to the current drawing.

Comment

Adds a comment to the signature.

Select time server

Sets a time server from the pop-up menu:



The selected time server will be used to obtain the precise time of the drawing signing. This means that when the signature is attached to the drawing during saving, the date and time from the selected time server will be inserted in the Signature date and time field of the Digital Signature Content. See the SIG-VALIDATE command.

9.45 -DIGITALSIGN command

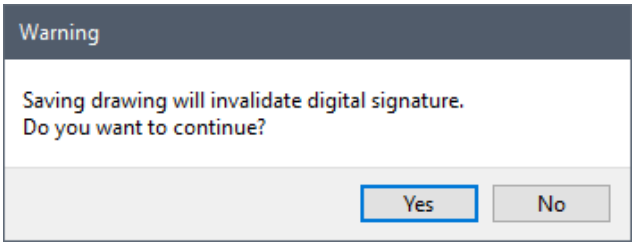
Applies a digital signature (block of encrypted information) to the drawing via Command line.

Shape Lite Pro Mechanical BIM

9.45.1 Method

Allows you to attach a digital signature to the drawing to validate the origin, authenticity, and unaltered state of the file since the digital signature was applied.

Note:

- The digital signature remains valid after renaming the drawing.
- After the -DIGITALSIGN command is launched, a digital signature is applied every time when saving the drawing until it is closed. The next time the signed drawing is opened and modified, a warning message appears when saving.
- 

9.45.2 Options within command

Attach digital signature after saving
Allows you to attach a digital signature to the drawing.

Yes
Applies a digital signature to the drawing.
Enter certificate serial number or select a certificate by its number in the list. Then, enter the time server name or select time server by its number in the list.

Note: The selected time server is used to obtain the precise time of the drawing signing. This means that when the signature is attached to the drawing during saving, the date and time from the selected time server is inserted in the Signature date and time field of the Digital Signature Content. See the SIGVALIDATE command.

No
Cancels the command.

9.46 DIM command

Creates multiple dimension types in a single workflow.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMENSION

9.46.1 Description

Allows you to create multiple dimensions or types of dimensions at once.

Note: Isometric view can be dimensioned and reflect the actual true geometry size.

9.46.2 Options within the command

HORizontal
Places horizontal linear dimensions, as does the DIMLINEAR command.

VErtical
Places vertical dimensions, as does the DIMLINEAR command.

ALigned
Places linear dimensions that are aligned to entities, as does the DIMALIGNED command.

ANgular
Places angular dimensions that measure angles, as does the DIMANGULAR command.

Leader
Places leaders, as does the DIMLEADER command.

OBlique
Changes the angle of extension lines, as does the DIMEDIT command.

ROtated
Places linear dimensions at an angle, as does the DIMLINEAR command.

CEnter
Places marks at the centers of circles and arcs, as does the DIMCENTER command.

Diameter
Places diameter dimensions on circles and arcs, as does the DIMDIAMETER command.

RAdius
Places radial dimensions on circles and arcs, as does the DIMRADIUS command.

Baseline
Places multiple linear and angular dimensions from the same base point, as does the DIMBASELINE command.

COntinue
Continues linear and angular dimensions from the last endpoint, same as the DIMCONTINUE command.

ORdinate
Places x and y ordinate measurements from an origin point, as does the DIMORDINATE command.

Position
Repositions the dimension text, as does the DIMTEDIT command.

DIStribute
Equally spaces the selected dimensions. There are two options to choose from on how the dimensions will be distributed.

Equal
Equally distributes all the selected dimensions.

Offset
All the selected dimensions are distributed at a specified offset distance.

UPdate dimensions
Applies the current dimension style to a selection of dimension entities; see the **Apply** option of the -DIMSTYLE command.

variable STatus
Lists in the **Prompt History** window the status of all dimension variables.

OVerride
Overrides the values of the current dimension style, as does the DIMOVERRIDE command.

SEttings...
Opens the **Drawing Explorer** | **Dimension Styles** dialog box which allows you to modify dimension styles, as does the DIMSTYLE command.

LAyer
Sets a different layer as the default layer on which the dimensions are drawn.

Note: The dimensions are created on the layer specified by the DIMLAYER system variable.

9.47 DIM1 Command

Executes a single dimension command at the 'Dimensioning command:' prompt.

Shape Lite Pro Mechanical BIM

9.47.1 Description

Allows you to create a single type of dimension, as specified in the Command line.

9.47.2 Options within the command

HORizontal
Places horizontal linear dimensions, as does the DIMLINEAR command.

VErtical
Places vertical dimensions, as does the DIMLINEAR command.

ALigned
Places linear dimensions that are aligned to entities, as does the DimAligned command.

ANgular
Places angular dimensions that measure angles, as does the DIMANGULAR command.

Leader
Places leaders, as does the DIMLEADER command.

OBlique
Changes the angle of extension lines, as does the DIMEDIT command.

ROtated
Places linear dimensions at an angle, as does the DIMLINEAR command.

CEnter
Places marks at the centers of circles and arcs, as does the DIMCENTER command.

Diameter
Places diameter dimensions on circles and arcs, as does the DIMDIAMETER command.

RAdius
Places radial dimensions on circles and arcs, as does the DIMRADIUS command.

Baseline
Places multiple linear and angular dimensions from the same base point, as does the DIMBASELINE command.

COntinue
Continues linear and angular dimensions from the last endpoint, as does the DIMCONTINUE command.

ORdinate
Places x and y ordinate measurements from an origin point, as does the DIMORDINATE command.

Position
Repositions the dimension text, as does the DIMTEDIT command.

DIStribute
Equally spaces the selected dimensions. There are two options to choose from on how the dimensions will be distributed.

Equal
Equally distributes all the selected dimensions.

Offset
All the selected dimensions are distributed at a specified offset distance.

UPdate dimensions
Applies the current dimension style to a selection of dimension entities; see the **Apply** option of the -DIMSTYLE command.

variable STatus
Lists in the **Prompt History** window the status of all dimension variables.

OVerride
Overrides the values of the current dimension style, as does the DIMOVERRIDE command.

SEttings...
Opens the **Drawing Explorer** | **Dimension Styles** dialog box which allows you to modify dimension styles, as does the DIMSTYLE command.

LAyer
Sets a different layer as the default layer on which the dimensions are drawn.

Note: The dimensions are created on the layer specified by the DIMLAYER system variable.

9.48 DIMALIGNED command

Creates an aligned dimension.

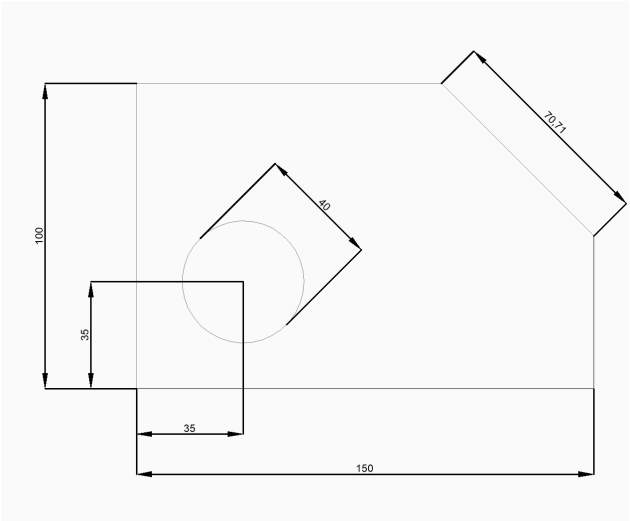
Shape Lite Pro Mechanical BIM

Icon: 

Alias: DAL, DIMALI

9.48.1 Description

Creates a dimension that aligns with the origin points of the extension lines. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



Note: Isometric view can be dimensioned and reflect the actual true geometry size.

9.48.2 Method

This command has 2 methods to begin creating an aligned dimension:

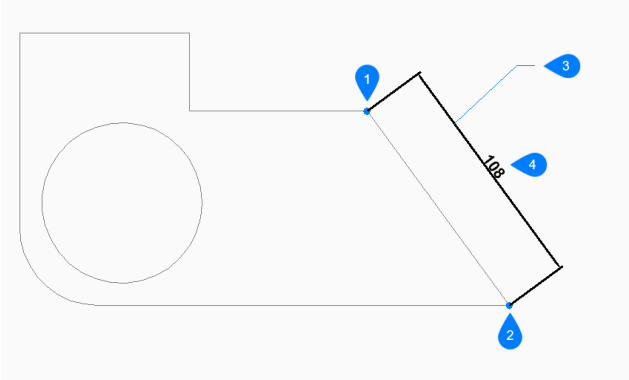
- Origin of first extension line
- Select entity

9.48.3 Options within the command

Origin of first extension line
Allows you to begin creating an aligned dimension by specifying a point for the first extension line.

Origin of second extension line
Allows you to specify a point for the second extension line.

Location of dimension line
Specifies the location of the dimension line. The dimension is placed at the same distance from each of the extension line origins.



1 Origin of first extension line

2 Origin of second extension line

3 Location of dimension line

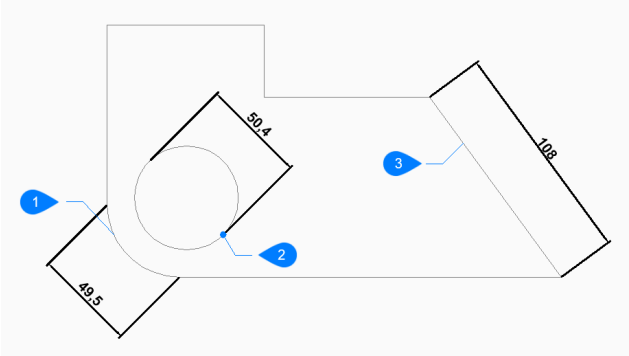
4 Aligned dimension

Select entity

Allows you to select a line, polyline segment, arc, or circle to dimension.

Note:

- When selecting a line, the endpoints of the line are used for the extension origins.
- When selecting an arc, the endpoints of the arc are used for the extension origins.
- When selecting a circle, the selection point on the circle is used for the first extension origin and the opposite point on the diameter of the circle is used for the second extension origin.



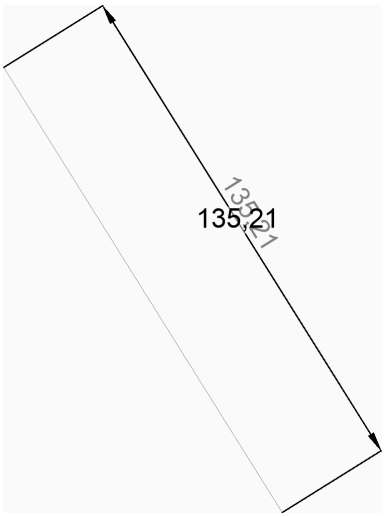
1 Arc entity

2 Selection point of circle entity

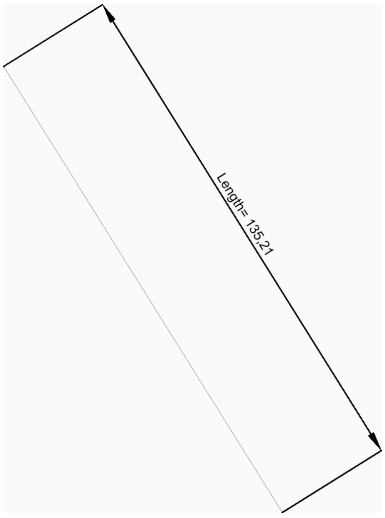
3 Polyline entity

Angle

Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text
Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.49 DIMANGULAR command

Creates an angular dimension.

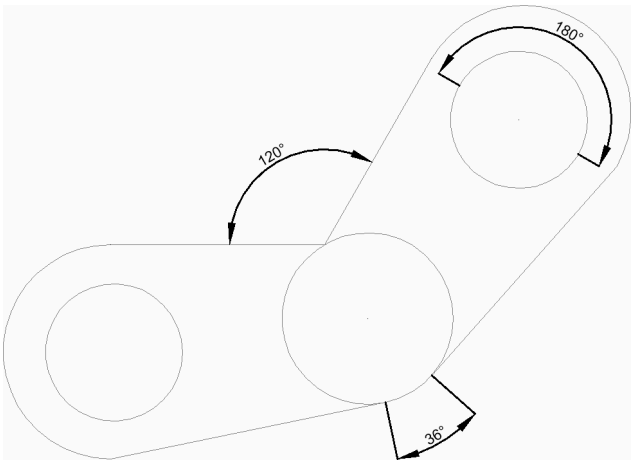
Shape Lite Pro Mechanical BIM

Icon:

Alias: DAN, DIMANG

9.49.1 Description

Creates an angular dimension by selecting an entity or specifying the vertex and both sides of the angle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.49.2 Method

This command has 2 methods to begin creating an angular dimension:

- Select line, arc, or circle
- Press ENTER to specify angle

9.49.3 Options within the command

Select line, arc or circle

Allows you to begin creating an angular dimension by selecting a line, arc, or circle to dimension.

If you selected a line or line segment the following option appear:

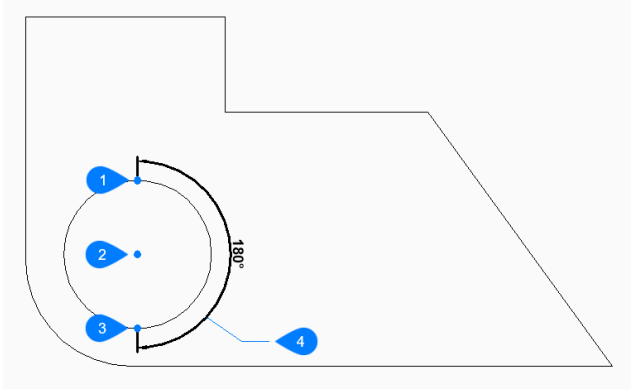
Other line for angular dimension

Allows you to specify another line segment.

If you selected a circle the following option appear:

Other side of angle

Allows you to specify a point to define one side of the angle. The point where you first selected the circle defines the other side of the angle.



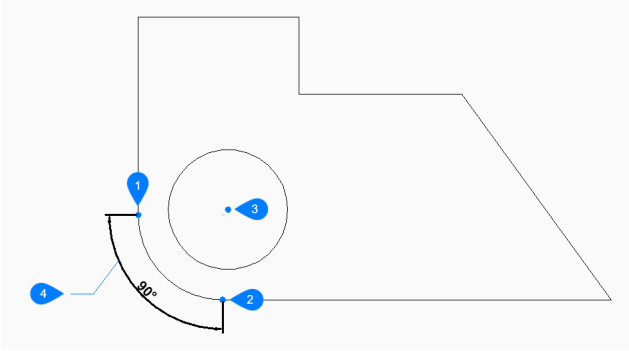
1 Select circle

2 Other side of circle

3 Vertex of angle

4 Location of dimension arc

Location of dimension arc
Specifies the location of the dimension arc.



1 First side of angle

2 Other side of angle

3 Vertex of angle

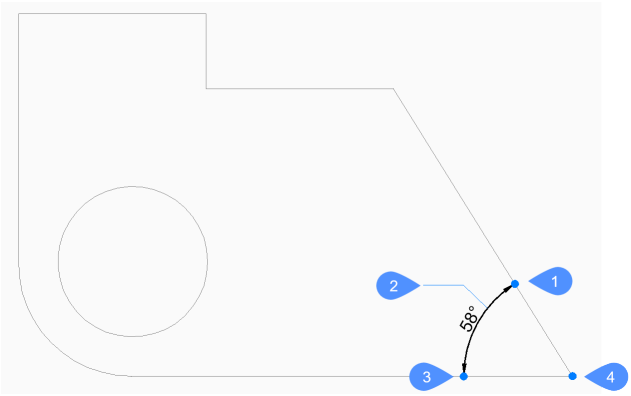
4 Location of dimension arc

Press ENTER to specify angle
Allows you to begin creating an angular dimension by specifying the vertex and the sides of the angle.

Vertex of angle
Allows you to specify the vertex of angle.

First side of angle
Specifies a point to define one side of the angle.

Other side of angle
Specifies a point to define the other side of the angle.



1 First side of angle

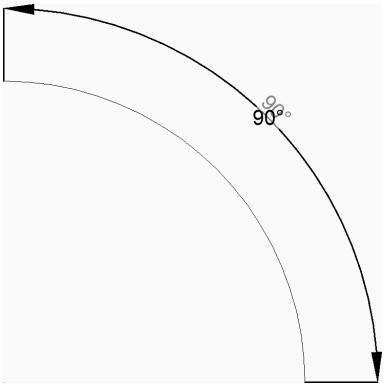
2 Location of dimension arc

3 Other side of angle

4 Vertex of angle

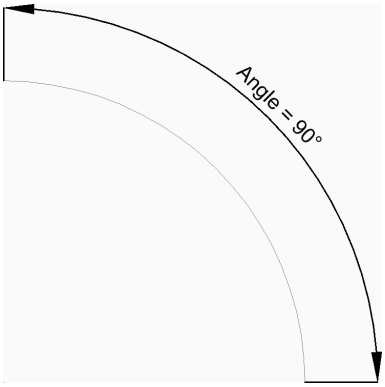
Angle

Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Allows you to enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.50 DIMARC command

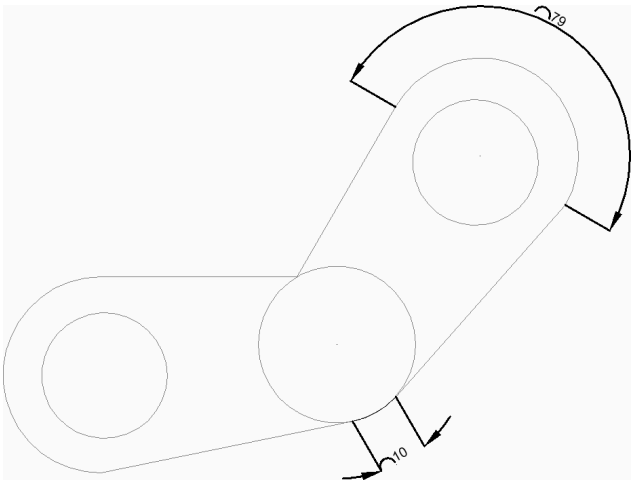
Creates an arc length dimension.

Shape Lite Pro Mechanical BIM

Icon:

9.50.1 Description

Creates a dimension that measure the length of an arc or polyarc. The dimension is based on the current dimension style. Options allow you to include a leader and specify the angle and contents of the dimension text.



9.50.2 Method

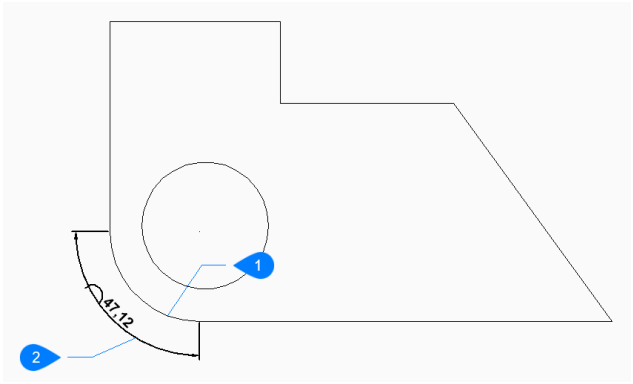
There is one method to begin creating an arc length dimension:

- Select arc or polyline arc segment

9.50.3 Options within the command

Select arc or polyline arc segment
Allows you to begin creating an arc length dimension by selecting an arc or polyline arc segment.

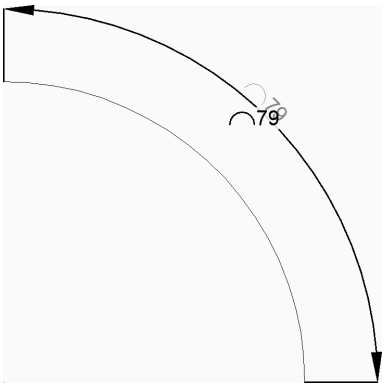
Location of dimension arc
Allows you to specify a point to locate the position of the dimension arc.



1 Arc segment

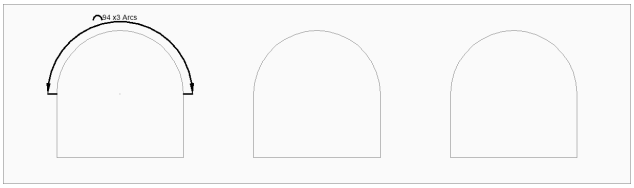
2 Location of dimension arc

Angle
Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



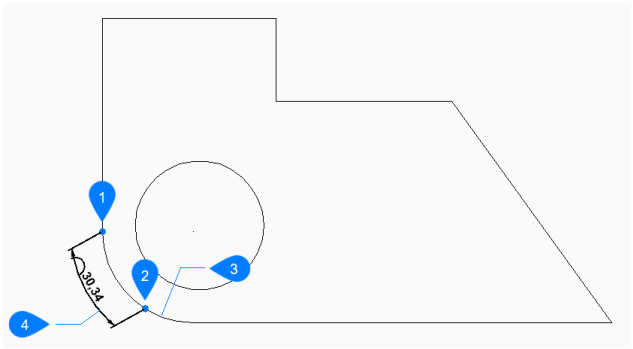
Text

Allows you to enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



Partial

Allows you to specify two points for the arc length dimension.



1 First point of arc length dimension

2 Second point of arc length dimension

3 Arc segment

4 Location of dimension arc

Leader

Toggles On the creation of a leader from the dimension text to the arc.

No leader

Toggle off the creation of the leader from the dimension text to the arc.

9.51 DIMBASELINE command

Creates stacked dimensions from the same baseline.

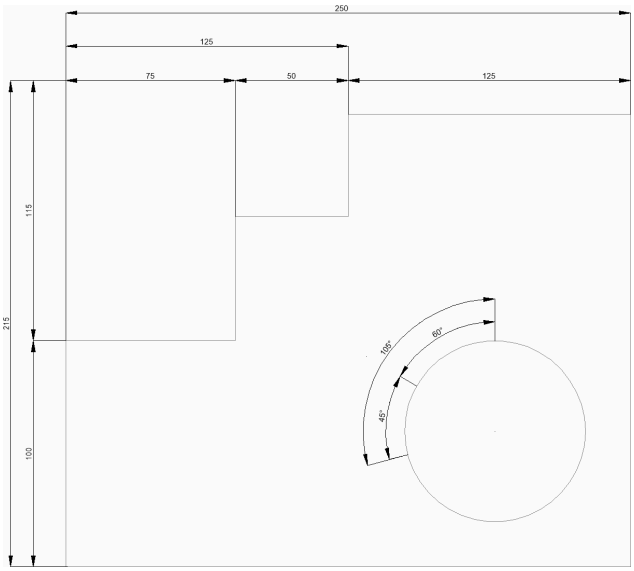
Shape Lite Pro Mechanical BIM

Icon:

Alias: DBA, DIMBASE

9.51.1 Description

Creates stacked linear, angular or ordinate dimensions from the same baseline as an existing dimension. The dimensions are based on the current dimension style and the dimension spacing is specified by the DIMDLI variable.



9.51.2 Method

This command has 2 methods to begin creating baseline dimensions:

- Origin of next extension line
- Select starting dimension

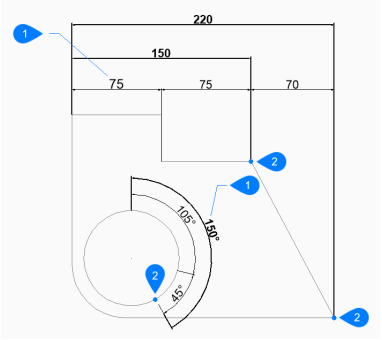
9.51.3 Options within the command

Origin of next extension line

Creates baseline dimensions from the last linear, angular, or ordinate dimension by specifying a point for the next extension line.

For linear and angular dimensions, the first extension line of each baseline dimension coincides with the first extension line of the previous dimension.

Note: Continue placing dimensions until you press Enter to end the command.



1 Existing dimension

2 Origin of next extension line

Select starting dimension
Allows you to select an existing linear, angular, or ordinate dimension.

Undo
Undoes the last dimension and continue drawing from the previous dimension.

9.52 DIMBREAK command

Breaks dimension lines, extension lines, and leaders at the locations where they cross other entities.

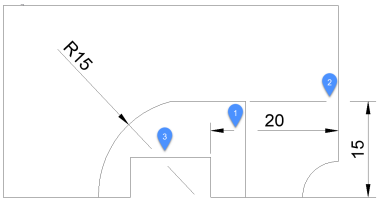
Shape Lite Pro Mechanical BIM

Icon:

9.52.1 Description

Allows breaking entities such as dimension lines, extension lines or leaders by a certain object and it can also remove breaks.

The dimension entity can be broken on different entities: the dimension line itself (1), on the extension line (2) or on the leader (3).



9.52.2 Options within the command

Multiple
Allows multiple dimension entities to be broken in the desired spots.

Auto
Breaks all the selected dimension entities at all intersections found with other entities.

Remove
Removes all dimension breaks from the selected dimension entities.

Manual
The width of the dimension break can be manually defined. This option is not available in Auto mode.

9.53 DIMCENTER command

Creates a center mark.

Shape Lite Pro Mechanical BIM

Icon:

9.53.1 Description

Create a center mark at the center of a circle, arc or polyarc. See the CENTERMARK command to create associative center marks.

9.53.2 Method

There is one method to begin creating a center mark.

- Select arc or circle to dimension

9.53.3 Options within the command

Select arc or circle to dimension
Creates a center dimension by selecting an arc, polyline arc segment, or circle.

Note: The DIMCEN system variable controls the length and appearance of the center mark lines.

9.54 DIMCONSTRAINT command

Applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Shape Lite Pro Mechanical BIM

9.54.1 Method

Select an associative dimension or else choose an option to place a dimensional constraint.

The associative dimension is converted to the dimensional constraint of the same type. This option is equivalent to the DCCONVERT command.

9.54.2 Options within the command

LInear
Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCLINEAR command.

Horizontal
Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCHORIZONTAL command.

Vertical
Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCVERTICAL command.

Aligned
Constrains the distance between two points. This option is equivalent to the DCALIGNED command.

ANgular
Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. This option is equivalent to the DCANGULAR command.

Radial
Constrains the radius of a circle or an arc. This option is equivalent to the DCRADIUS command.

Diameter
Constrains the diameter of a circle or an arc. This option is equivalent to the DCDIAMETER command.

9.55 DIMCONTINUE command

Creates dimensions in a continuous line or arc.

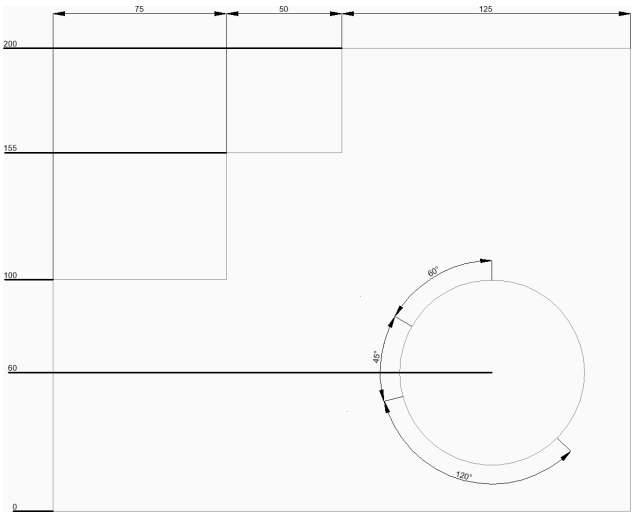
Shape Lite Pro Mechanical BIM

Icon:

Alias: DCO, DIMCONT

9.55.1 Description

Creates a continuous line or arc from an existing linear, angular, or ordinate dimension. The dimensions are based on the current dimension style.



9.55.2 Method

This command has 2 methods to begin creating continuous dimensions:

- Origin of next extension line
- Select starting dimension

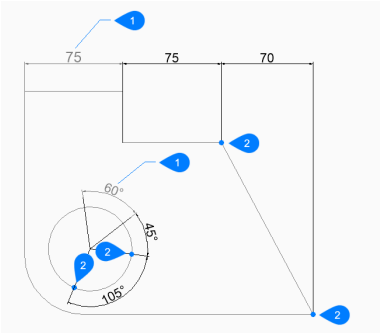
9.55.3 Options within the command

Origin of next extension line

Creates continuous dimensions from the last linear, angular, or ordinate dimension by specifying a point for the next extension line.

For linear and angular dimensions, the first extension line of each continuous dimension coincides with the second extension line of the previous dimension.

Note: Continue placing dimensions until you press Enter to end the command.

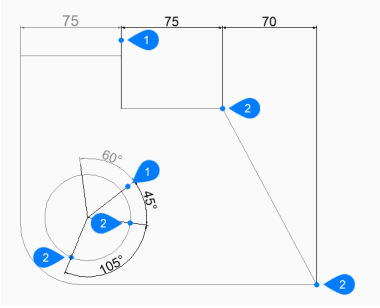


1 Last dimension

2 Origin of next extension line

Select starting dimension

Allows you to select an existing linear, angular, or ordinate dimension.



1 Starting dimension

2 Origin of next extension line

Undo

Undoes the last dimension and continue drawing from the previous dimension.

9.56 DIMDIAMETER command

Creates a diametric dimension.

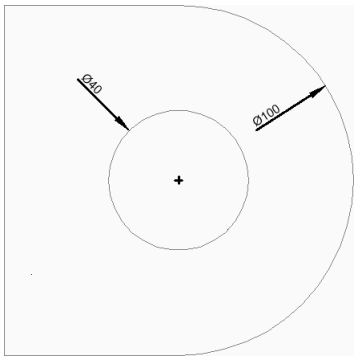
Shape Lite Pro Mechanical BIM

Icon:

Alias: DDI, DIMDIA

9.56.1 Description

Creates a diametric dimension for an arc, polyarc, or circle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.56.2 Method

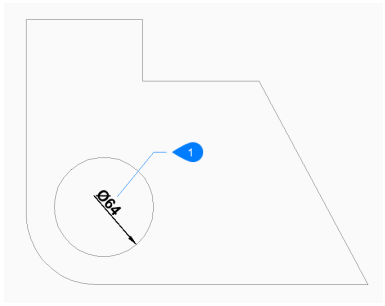
There is one method to begin creating a diametric dimension:

- Select arc or circle to dimension

9.56.3 Options within the command

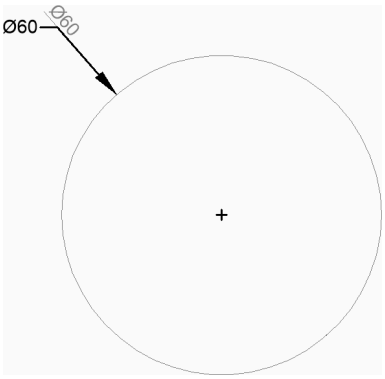
Select arc or circle to dimension
Allows you to begin creating a diametric dimension by selecting an arc, polyline arc, or circle.

Location of dimension line
Specifies the location of the dimension line.



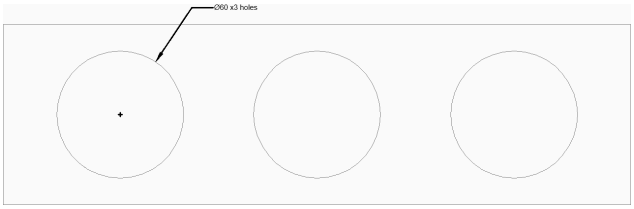
1 Location of dimension line

Angle
Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Allows you to enter text to override the default dimension text which includes the measured length of the diameter. You can use two angle brackets <> to display the measured length of the diameter in addition to other text.



9.57 DIMDISASSOCIATE command

Removes the associativity of selected dimension entities.

Shape Lite Pro Mechanical BIM

Icon:

9.57.1 Description

Disassociates the selected dimension entities and prompts you in the Command line the number of disassociated dimensions.

9.58 DIMEDIT command

Edits particular dimension elements.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DED, DIMED

9.58.1 Description

Edits the position, angle, and wording of dimension text, and changes the angle of extension lines.

9.58.2 Methods

There are four methods to edit dimensions:

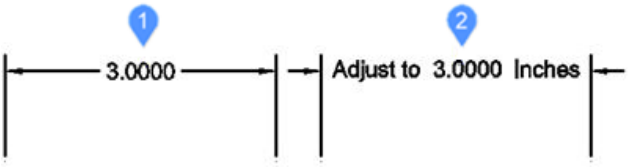
- Edit text
- Oblique lines
- Rotate text
- Restore text

9.58.3 Options within the command

Edit text

This option will allow you to edit the existing dimension text.

Use any selection mode to choose the dimension entities to be edited. Use the <> symbols to indicate the default text (1), so that you can insert text in front or behind the default, such as “Adjust to <> inches” (2).

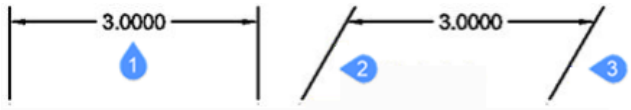


Oblique lines

This option will rotate (obliques or slants) the extension lines of selected linear dimensions.

The obliquing angle is measured counterclockwise from the positive x-axis.

Select linear dimensions (1) and enter the obliquing angle (2). The extension lines (3) will become oblique.

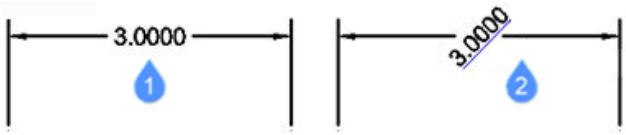


Rotate text

This option will rotate the dimension text.

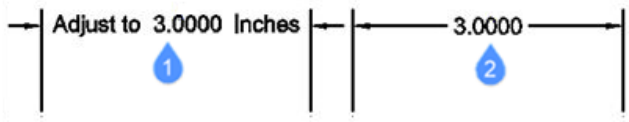
The dimension text rotation angle is measured counterclockwise from the positive x-axis.

Select the dimension text (1) and enter the angle of dimension text (2).



REstore text

This option will restore the dimension text (1) back to its original position (2). This option does not restore edited text or obliqued extension lines.



9.59 DIMEX command (Express Tools)

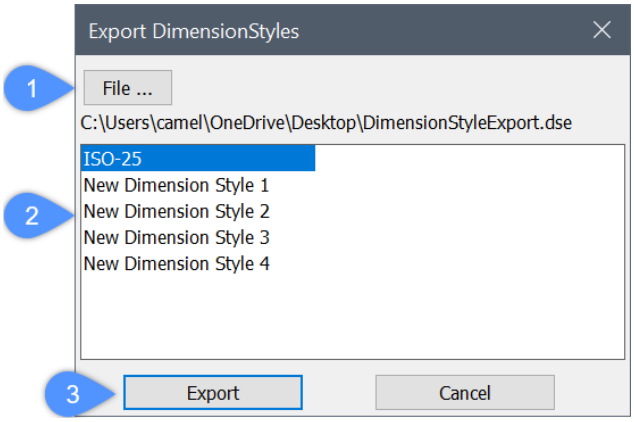
Exports dimension styles and their settings to an external file.

Shape Lite Pro Mechanical BIM

Icon:

9.59.1 Method

Opens the **Export DimensionStyles** dialog box that allows you to export the available dimension styles from the current drawing to a DSE. The file can be imported in another drawing using the DIMIM command.



1 File...

2 Available Dimension Styles

3 Export

9.59.2 File...

Opens the **Select export file** dialog box that allows you to choose the location for the DSE file.

9.59.3 Available Dimension Styles

Displays the available dimension styles and allows you to select the dimension styles to write to the DSE file. The selected styles are written in the file indicated under the **File...** button.

9.59.4 Export

Exports the selected dimension styles to the indicated DSE file.

9.60 DIMIM command (Express Tools)

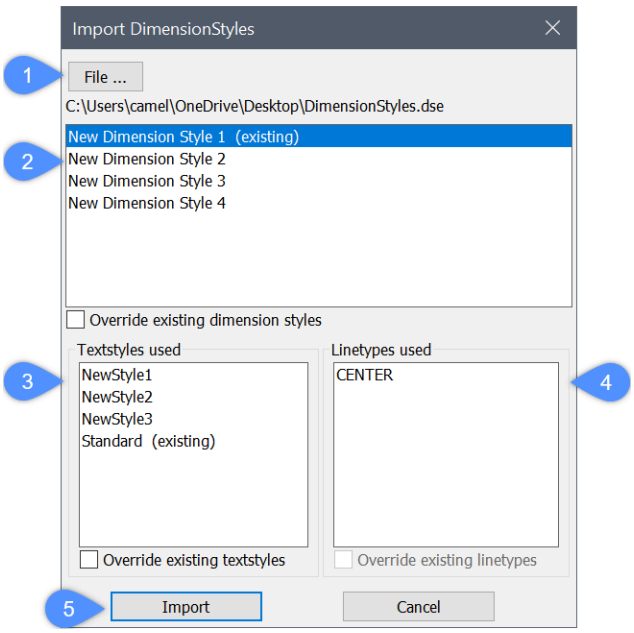
Imports dimension styles and their settings from an external file.

Shape Lite Pro Mechanical BIM

Icon:

9.60.1 Method

Opens the **Import DimensionStyles** dialog box that allows you to import dimension styles that were previously exported using the DIMEX command.



1 File...

2 Dimension styles used

3 Textstyles used

4 Linetypes used

5 Import

9.60.2 File...

Opens the **Select import file** dialog box that allows you to choose a DSE file to import. All dimension styles in the selected file are imported into the current drawing, including the text styles and linetypes.

Note: The DSE file must have the same format as that generated by the DIMEX command.

9.60.3 Dimension styles used

Displays the dimension styles from the DSE file and allows you to select the dimension styles to import into the current drawing.

Note: Tick the **Override existing dimension styles** option to overwrite the dimension styles in the current drawing and use the similarly named dimension style from the DSE file.

9.60.4 Textstyles used

Displays the text styles from the DSE file.

Note: Tick the **Override existing textstyles** option to overwrite the text styles in the current drawing and use the similarly named text style from the DSE file.

9.60.5 Linetypes used

Displays the linetypes from the DSE file.

Note: Tick the **Override existing linetypes** option to overwrite the linetypes in the current drawing and use the similarly named linetypes from the DSE file.

9.60.6 Import

Imports the selected dimension styles from the DSE file.

9.61 DIMJOGGED command

Creates a jogged dimension for arcs or circles.

Shape Lite Pro Mechanical BIM

Icon:

9.61.1 Description

Creates a jogged dimension for arcs or circles. It is useful when the center of the arc or circle is outside the layout area.



9.61.2 Method

Select an arc or a circle to add a jogged dimension and specify the center location override.

9.61.3 Options within the command

Angle
Sets an angle of the dimension text.

Text
Sets the dimension text that represents the actual dimension.

9.62 DIMJOGLINE command

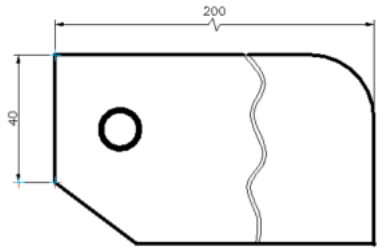
Applies or removes a jog line to a linear or aligned dimensions.

Shape Lite Pro Mechanical BIM

Icon:

9.62.1 Description

Applies a jog line to linear or aligned dimensions. The dimension should represent the actual dimension.



9.62.2 Method

Select the linear or angular dimension to add a jog and specify the jog location.

9.62.3 Options within the command

Remove
Removes the jog of a linear or alined dimension.

9.63 DIMLEADER command

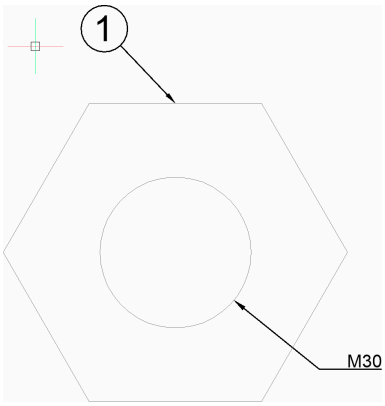
Creates a leader.

Shape Lite Pro Mechanical BIM

Icon:

9.63.1 Description

Creates a leader by specifying a sequence of points. The leader is based on the current dimension style. Options allow you to specify the format and annotation of the leader.



9.63.2 Method

There is one method to begin creating a leader:

- Start of leader

9.63.3 Options within the command

Start of leader
Allows you to begin creating a leader by specifying a start point.

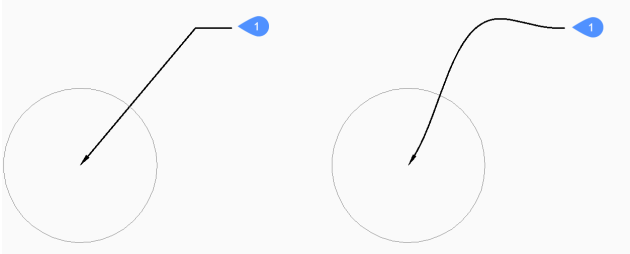
Next point
Specifies the next vertex of the leader.

To point
Specifies the next vertex.

Note: You can continue adding unlimited vertices until you press Enter to access the annotation option.

Format
Specifies if the leader line includes an arrow and whether it has spline or straight segments:

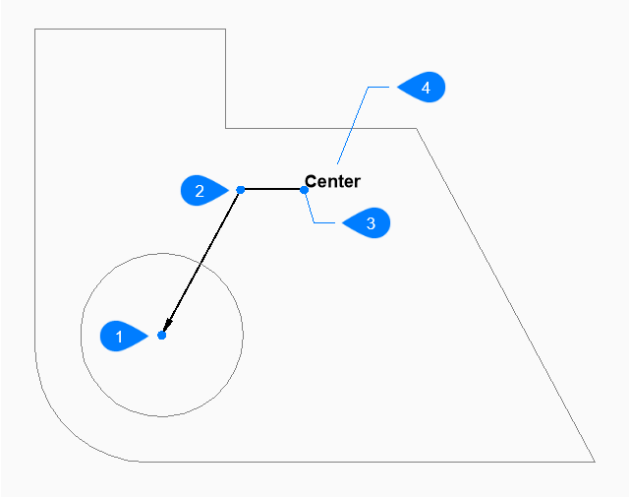
- **Arrow**: draw the arrowhead.
- **None**: does not draw the arrowhead.
- **SPline**: draw the leader line as a spline.
- **STraight**: draw the leader line as straight segments.
- **Exit**: exit the Format options.



1 Center

Undo
Undoes the last leader line segment and continue drawing from the previous one.

Annotation
Allows you to type the lines of annotation text.



1 Start of leader

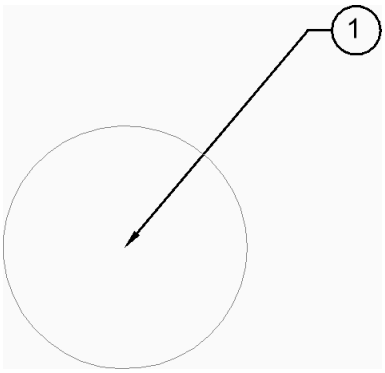
2 Next point (vertex)

3 To point

4 Annotation

options
Allows you to set the dimension text options.

Block
Specifies the name of a block in the drawing.



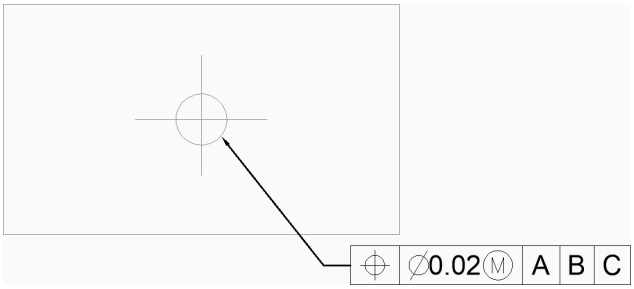
? to list blocks in drawing
Enter * to list the names of all block definitions in the current drawing. You can also use * as a wildcard with other characters.

Enter **~** to open the **Insert Block** dialog box that allows you select a DWG file to use as an annotation block.

Copy
Allows you to select an mtext, text, block reference or tolerance entity in the drawing.

None
Creates the leader line without annotations.

Tolerance
Specifies the tolerance using the **Geometric Tolerance** dialog box.



MText
Allows you to enter the annotation text using the MText editor.

9.64 DIMLINEAR command

Creates a linear dimension.

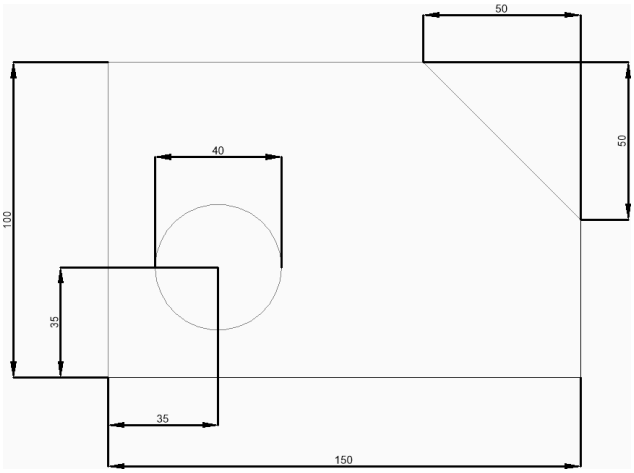
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMHORIZONTAL, DIMLIN, DIMROTATED, DIMVERTICAL, DLI

9.64.1 Description

Creates a linear dimension that is horizontal, vertical, or rotated. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.64.2 Method

This command has 2 methods to begin creating a linear dimension:

- Origin of first extension line
- Select entity

9.64.3 Options within the command

Origin of first extension line

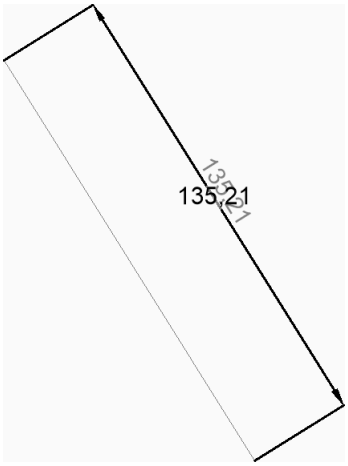
Allows you to begin creating a linear dimension by specifying a point for the first extension line.

Origin of second extension line

Specifies a point for the second extension line.

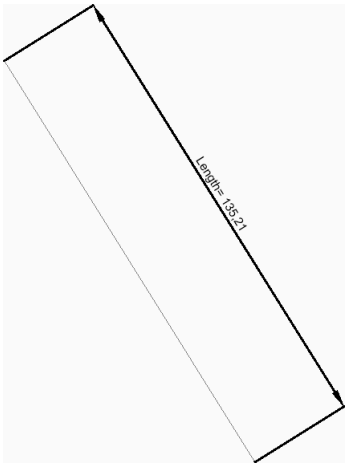
Angle

Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Allows you to enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



Horizontal

Specifies the location of the dimension to place it in the drawing. This creates a horizontal dimension regardless of how the endpoints are aligned.

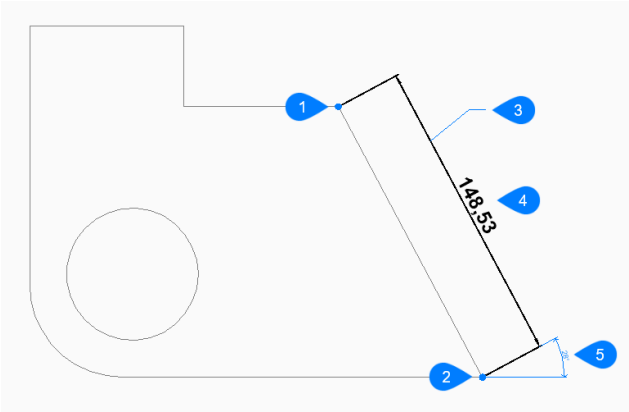
Vertical

Specifies the location of the dimension to place it in the drawing. This creates a vertical dimension regardless of how the endpoints are aligned.

Rotated

Specifies an angle of the dimension line.

Note: The dimension is rotated from its default (vertical or horizontal) location by the specified angle.



1 Origin of first extension line

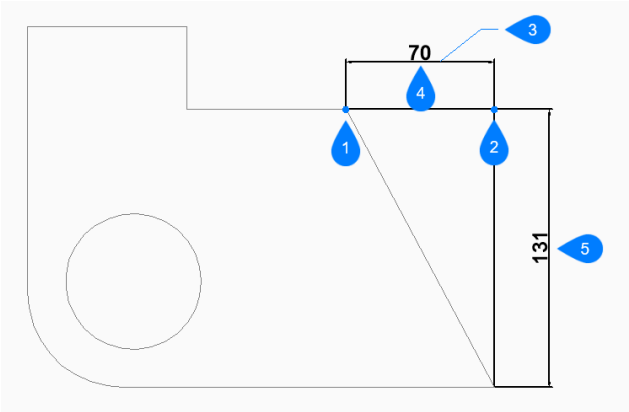
2 Origin of second extension line

3 Location of dimension line

4 Rotated dimension

5 Angle of dimension line

Specify the location of the dimension line. If the two extension points you specified are aligned vertically or horizontally, you are restricted to placing a vertical or horizontal dimension, respectively. If the two extension points you specified are not vertically or horizontally aligned, you can drag the cursor to place either a vertical or a horizontal dimension.



1 Origin of first extension line

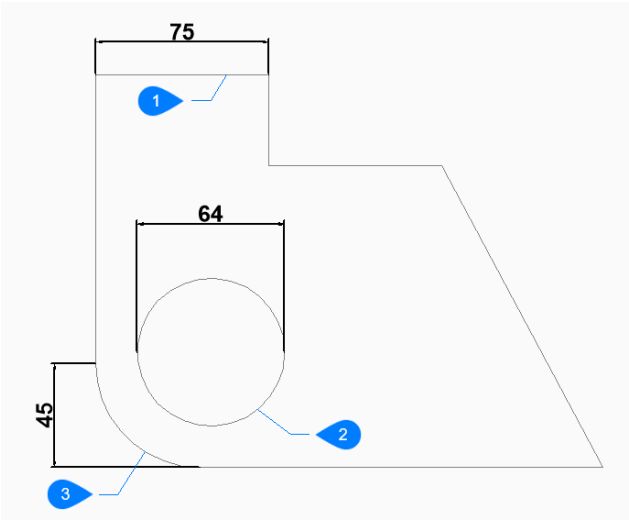
2 Origin of second extension line

3 Location of dimension line

4 Horizontal dimension

5 Vertical dimension

Select entity
Allows you to select an entity (line, polyline segment, arc, or circle) to dimension.
When selecting a line, the endpoints of the line are used for the extension origins.
When selecting an arc, the endpoints of the arc are used for the extension origins. When selecting a circle, the quadrants of the circle are used for the extension origins.



1 Line entity

2 Circle entity

3 Arc entity

9.65 DIMMARKOVERRIDES command

Adds or removes underline to dimensions with overridden dimtext.

Shape Lite Pro Mechanical BIM

9.65.1 Description

Adds or removes underline to dimensions with overridden dimtext by making database modifications.

Select dimensions
Select dimensions with overridden dimension text. When all desired dimensions are selected, press **Enter**.

Note: To mark dimensions with overridden dimtext without database modification, set the DIMMARKTYPE system variable value to 1 or 2.

9.65.2 Options within the command

Once all desired entities are selected, two options are available:

Mark
Underlines the overridden text for the selected dimensions.

Unmark
Removes the overridden text underline for the selected dimensions.

9.66 DIMORDINATE command

Creates an ordinate dimension.

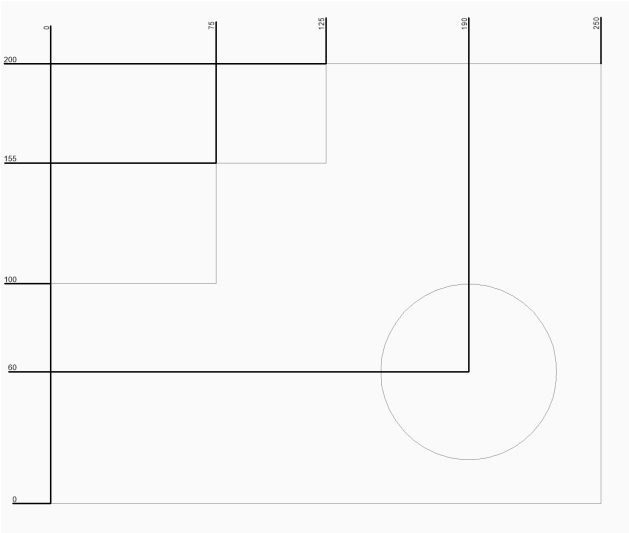
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMORD, DOR

9.66.1 Description

Create an ordinate dimension that measures the x or y distance from the current UCS origin to a specified point in the drawing. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.66.2 Method

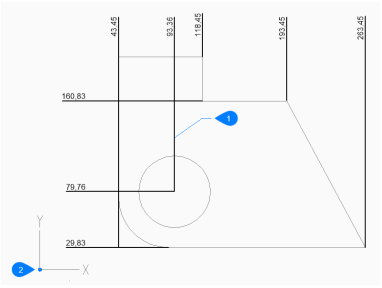
This command has 1 method to begin creating an ordinate dimension:

- Select point for ordinate dimension

9.66.3 Options within the command

Select point for ordinate dimension
Allows you to begin creating an ordinate dimension by specifying a start point for the leader.

Leader endpoint
Specifies an endpoint for the leader. The direction you drag from the start point determines whether you create an X or Y ordinate.



1 Location of ordinate line

2 UCS Basepoint

Xdatum

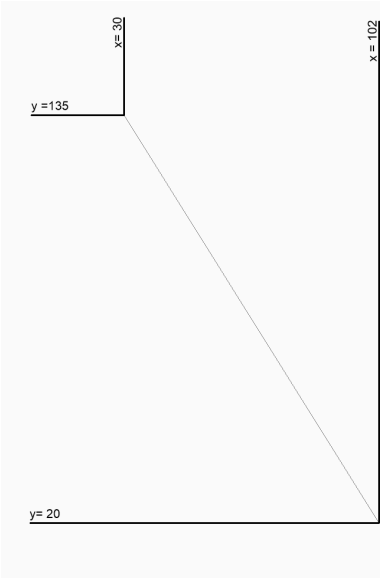
Specifies the endpoint for an X ordinate regardless of which direction you drag from the start point.

Ydatum

Specifies the endpoint for a Y ordinate regardless of which direction you drag from the start point.

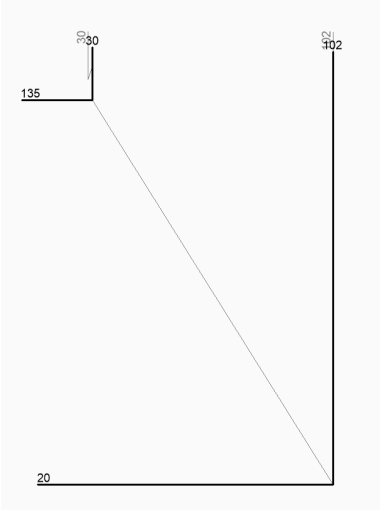
Text

Allows you to enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



Angle

Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



9.67 DIMOVERRIDE command

Overrides the values of the current dimension style.

Shape Lite Pro Mechanical BIM

Alias: DIMOVER, DOV

9.67.1 Description

Allows the override of dimension style values in a selected dimension.

Note: This command is superseded by the Override function of the DIMSTYLE command.

9.67.2 Options within the command

Clear

Resets the values of dimension variables, clearing the overrides.

9.68 DIMRADIUS command

Creates a radial dimension.

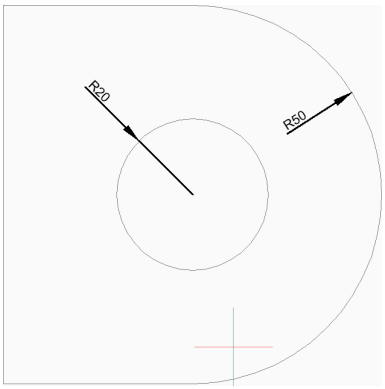
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMRAD, DRA

9.68.1 Description

Creates a radial dimension for an arc, polyarc, or circle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.68.2 Method

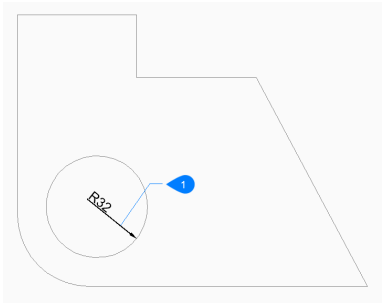
This command has 1 method to begin creating a radial dimension:

- Select arc or circle to dimension

9.68.3 Options within the command

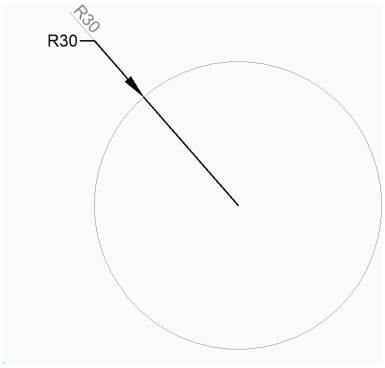
Select arc or circle to dimension
Allows you to begin creating a radial dimension by selecting an arc, polyline arc, or circle.

Location of dimension line
Specifies the location of the dimension line.



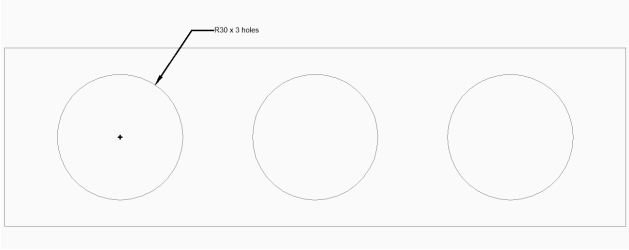
1 Location of dimension line

Angle
Specifies the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Allows you to enter text to override the default dimension text which includes the measured length of the radius. You can use two angle brackets <> to display the measured length of the radius in addition to other text.



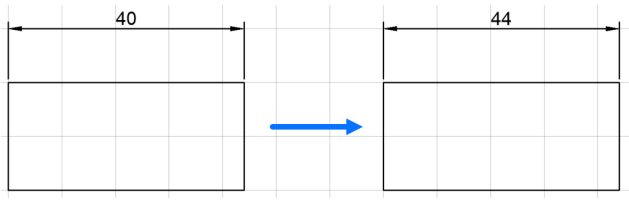
9.69 DIMREASSOC command (Express Tools)

Restores the measured value of a dimension text that has been previously modified or overwritten.

Shape Lite Pro Mechanical BIM

9.69.1 Method

Select entities with non-associative text and press Enter.



9.70 DIMREASSOCIATE command

Reassociates or associates dimensions to entities or points on entities.

Shape Lite Pro Mechanical BIM

Icon:

9.70.1 Description

Selected dimension entities highlight one by one with prompts to specify association points or entities for the currently highlighted dimension entity.

Note: A marker displays at the current position of the edited dimension point:

- indicates a non-associated dimension point.
- indicates an associated dimension point.

9.70.2 Methods

The DIMREASSOCIATE command can be used in six different ways:

- Linear (rotated and aligned): specify an entity or two lines.
- Diameter: select arc, circle or polyline arc.

- Radius: select arc, circle or polyline arc.
- Angular: select 2 lines or 3 points, an Arc or a circle.
- Ordinate: specify feature location.
- Leader: specify leader association point.

9.70.3 Options

Disassociated
Prompts to reassociate ALL disassociated dimension entities in the drawing. All disassociated dimensions highlight one by one.

9.71 DIMREGEN command

Updates associative dimensions.

Shape Lite Pro Mechanical BIM

9.71.1 Description

Updates ALL associative dimensions in the drawing file.

9.72 DIMSPACE command

Adjusts the spacing between the lines of parallel linear/angular dimensions.

Shape Lite Pro Mechanical BIM

9.72.1 Description

Allows you to equally adjust the spacing between the lines of parallel linear/angular dimensions that share a common vertex, starting from a base dimension.

9.72.2 Method

There are two methods to adjust parallel linear/angular dimensions spacing:

- equally space dimensions lines
- line up a series of dimensions lines

9.72.3 Options within the command

Select base dimension
Allows you to select the base dimension from which the other dimensions will be spaced.

Select dimensions to space
Allows you to select dimensions to space equally/line up from the base dimension, then press Enter.

Enter value
Allows you to enter a spacing value.

Note: To line up the dimension lines, enter a spacing value of 0.

Auto
The spacing distance becomes twice the text height specified in the dimension style of the selected base dimension.

9.73 DIMSTYLE command

Opens the **Drawing Explorer** dialog box with **Dimension Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

Alias: D, DDIM, DIMSTY, DS, DST, EXPDIMSTYLES, SETDIM

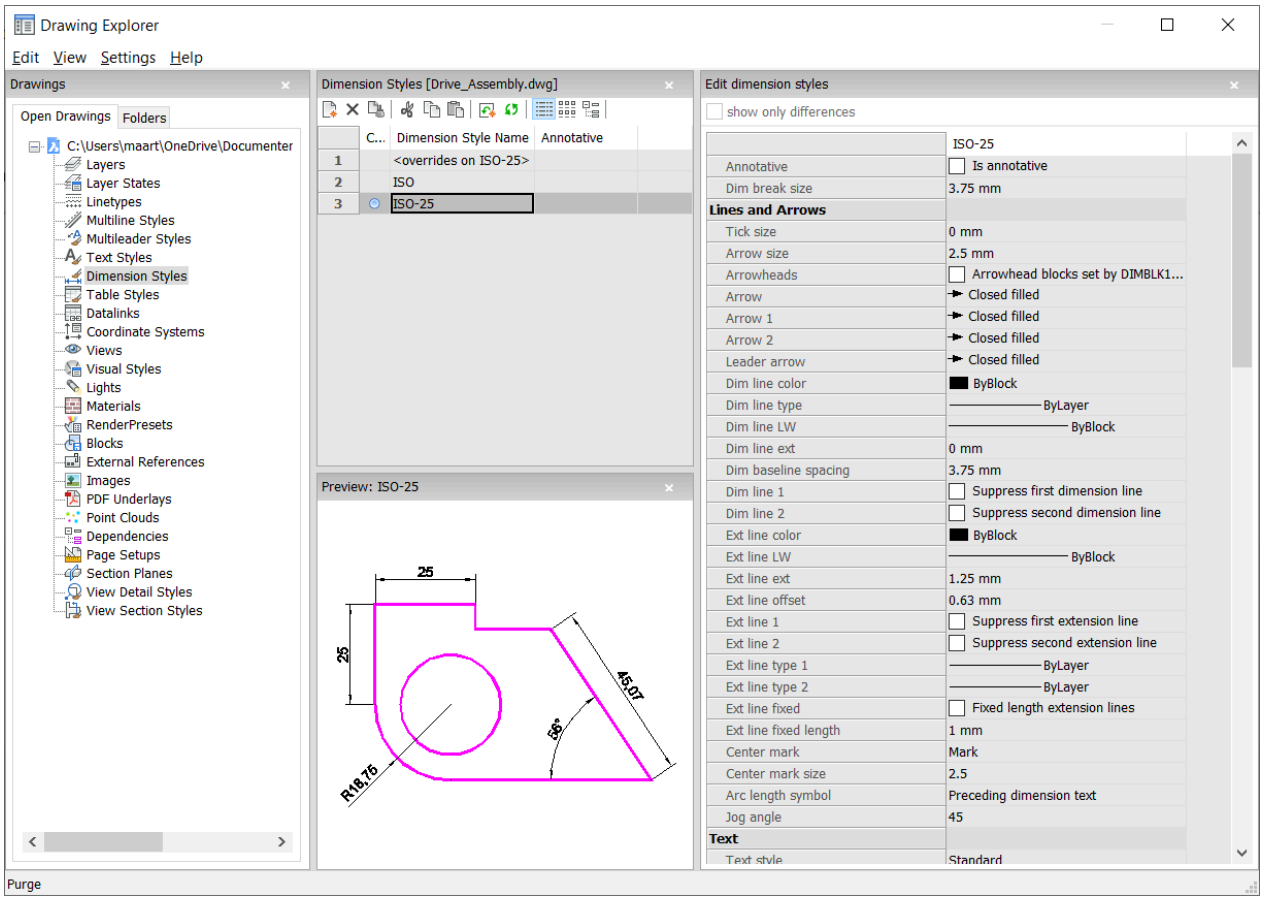
9.73.1 Description

Opens the **Drawing Explorer** dialog box with **Dimension Styles** category selected to view and modify dimension styles in the current drawing.

9.73.2 Method

There are two methods when using DIMSTYLE command:

- <Overrides>: overrides the values of individual dimension variables without creating a new style.
- Standard: lists the dimension settings for the dimension style named "Standard".



9.73.3 Context Menu Options

The following options appear when you right-mouse click on a dimension style:

New
Loads additional Dimension Styles definitions into the drawing.

Delete
Deletes Dimension Styles definitions from the drawing. The following dimension styles cannot be deleted:

- <overrides on ISO-25>/<overrides on Standard>
- ISO-25/Standard

Rename
Renames the Dimension Styles. The following dimension styles cannot be renamed:

- <overrides on ISO-25>
- ISO-25

Select All
Selects all Dimension Styles definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected Dimension Style as current.

Save overrides to current style
Saves the override variables of the selected dimension style to the current dimension style.

Save to new style
Creates a new dimension style based on the selected dimension style.

New child style
Creates a new dimension child style. A dimension style can have up to 6 child styles: Linear, Angular, Radius, Diameter, Ordinate, and Leader. If defined, then the child style is used for the corresponding dimension type.
Dimension child styles share all settings with the parent style, except for the properties that are defined explicitly differently.

9.73.4 Options within the Edit dimension styles panel

Show only differences
If multiple styles are selected, compares the selected styles, and shows the differences only.

Annotative
Sets the annotative property of the dimension style.

Dim break size
Sets the value of the gap created by the DIMBREAK command.

Lines and Arrows

Lines and Arrows	
Tick size	0 mm
Arrow size	2.5 mm
Arrowheads	Arrowhead blocks set by DIMBLK1...
Arrow	Closed filled
Arrow 1	Closed filled
Arrow 2	Closed filled
Leader arrow	Closed filled
Dim line color	ByBlock
Dim line type	ByLayer
Dim line LW	ByBlock
Dim line ext	0 mm
Dim baseline spacing	3.75 mm
Dim line 1	Suppress first dimension line
Dim line 2	Suppress second dimension line
Ext line color	ByBlock
Ext line LW	ByBlock
Ext line ext	1.25 mm
Ext line offset	0.63 mm
Ext line 1	Suppress first extension line
Ext line 2	Suppress second extension line
Ext line type 1	ByLayer
Ext line type 2	ByLayer
Ext line fixed	Fixed length extension lines
Ext line fixed length	1 mm
Center mark	Mark
Center mark size	2.5
Arc length symbol	Preceding dimension text
Jog angle	45

Tick size

Determines the size of tick marks drawn instead of arrowheads for linear, radius and diameter dimensioning. If the value is zero, arrowheads are drawn.

Arrow size

Determines the size of arrowheads of dimension lines and leader lines.

Arrowheads

Controls whether dimension line arrowhead blocks are set by DIMBLK or DIMBLK1 or DIMBLK2.

Arrow

Determines the name of the block displayed at the ends of dimension lines and leader lines. The block name can be either a standard name or refer to a user-defined arrowhead block.

Arrow 1

Specifies the style of arrowhead for the starting end of the dimension line.

Arrow 2

Specifies the style of arrowhead for the other end of the dimension line.

Leader arrow

Specifies the style of arrowhead for the starting end of the leader line.

Dim line color

Specifies the color of the dimension line; choose from:

- Any Index Color
- Any True Color

- ByBlock
- ByLayer

Dim line type
Specifies the line type of the dimension line. Choose from any line type loaded into the current drawing. To access additional linetypes, click Load..., and then choose one from the **Load Linetypes** dialog box. See the LINETYPE command.

Dim line LW
Specifies the lineweight of the dimension line. Choose from any lineweight supported by the program.

Dim line ext
Specifies the distance that the dimension line extends beyond the extension lines.

Dim baseline spacing
Specifies the default distance between dimensions added with the DIMBASELINE command.

Dim line 1
Toggles the display of the first half of the dimension line (between the starting extension line and the text).

Dim line 2
Toggles the display of the second half of the dimension line (between the other extension line and the text).

Ext line color
Specifies the color of the dimension line. To select additional colors, click Select color... and choose one from the **Color** dialog box.
See the COLOR command.

Ext line LW
Specifies the lineweight of the extension line. Choose from any lineweight supported by the program.

Ext line ext
Specifies the distance that the extension lines extend beyond the dimension line.

Ext line offset
Specifies the offset distance between the object and the start of the extension lines.

Ext line 1
Toggles the display of the first extension line.

Ext line type 1
Specifies the line type of the first extension line. You can choose from any line type loaded into the current drawing.
To access additional linetypes, click Load... . The **Load Linetypes** dialog box is displayed where you can choose a linetype.

Ext line type 2
Specifies the line type of the second extension line. You can choose from any line type loaded into the current drawing.
To access additional linetypes, click Load... . The **Load Linetypes** dialog box is displayed where you can choose a linetype.

Ext line 2
Toggles the display of the second extension line.

Ext line fixed
Determines whether fixed length extension lines are used.

Ext line fixed length
Specifies the total length of the extension lines.

Center mark
Specifies the type of center mark:

- Mark
- Line
- None

Center mark size
Specifies the size of the center mark.

Arc length symbol
Specifies the location of the arc length symbol:

- Preceding dimension text.
- Above dimension text.
- Not displayed.

Jog angle
Specifies the angle of the jog (default is 45 degrees). Enter another angle.

Text

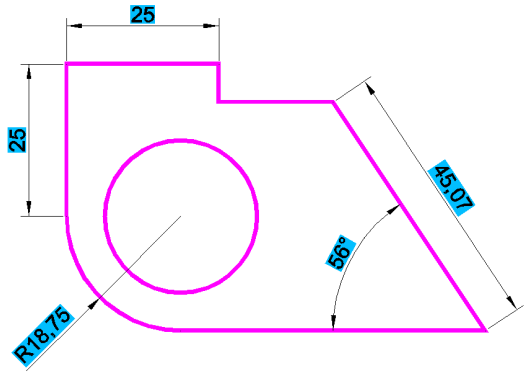
Text	
Text style	Standard
Text color	ByBlock
Text fill	No fill
Text fill color	ByBlock
Text height	2.5 mm
Draw frame around text	Draw frame around text
Text position vertical	Above
Text position horizontal	Centered
Text offset	0.625
Text vertical offset	0
Text inside align	Horizontal
Text outside align	Horizontal
Text view direction	Left to right

Text style
Specifies the style for dimension text; can use only styles created with the STYLE command.

Text color
Specifies the color of the text. Click **Select color...** to select additional color from the **Color** dialog box.

Text fill
Determines whether the dimension text has a rectangular background filled with color.

- No fill: does not apply a background fill.
- Background: uses the background color of the drawing, usually white or black.
- Color: uses the color specified by the Text Fill Color option.



Text fill color
Specifies the background fill color when Text Fill is set to Color. You can choose a color from the drop-down list or choose **Select color...**, and choose a color from the **Color** dialog box.

Text height
Specifies the height of the text.

Draw frame around text
Draws a rectangle around the text.

Text position vertical
Justifies the text vertically relative to the dimension line:

- **Centered**: centers the text on the dimension line.
- **Above**: places the text above the dimension line.
- **Outside**: places text outside of the extension lines.
- **JIS**: places text above the dimension line, according to the Japanese Industry Standard.
- **Below**: places the text below the dimension line.

Text position horizontal
Justifies the text horizontally relative to the extension lines:

- **Centered**: centers the text between the extension lines.
- **First Extension Line**: places the text near the first extension line.
- **Second Extension Line**: places the text near the second extension line.
- **Over First Extension**: places the text over the first extension line.
- **Over Second Extension**: places the text over the second extension line.

Text offset
Specifies the size of gap between the dimension line and the text.

Text vertical offset
Sets the vertical position of dimension text above or below the dimension line.

Text inside align
Justifies text when located between extension lines:

- **Aligned with Dimension Line**: aligns text with dimension line.
- **Horizontal**: forces text to be always horizontal.

Text outside align

Justifies text when located outside of the extension lines:

- **Aligned with Dimension Line**: aligns text with dimension line.
- **Horizontal**: forces text to be always horizontal.

Text view direction

Sets the dimension text reading direction.

Fit

Fit	
Arrow and text fit	Best fit
Text inside	☐ Force text between extension lines
Dim line inside	☐ Suppress outside arrowheads
Text movement	Keep dim line with text
Dim scale overall	1
Place text manually	☐ Cursor controls both the text posit...
Dim line forced	☑ Force dimension lines even when ...

Arrow and text fit

Specifies where text and arrows should be located when there is insufficient space for both between the extension lines:

- **Text and Arrows**: forces both text and arrows between the extension lines.
- **Arrows Only**: forces arrows to stay within extension lines; moves text outside when there is insufficient space.
- **Text Only**: forces text to stay within extension lines; moves arrows outside when there is insufficient space.
- **Best Fit**: requires the program to figure out where to place text and arrows, depending on the space between extension lines.

Text inside

Specifies whether the text is forced between extension lines:

- On: forces text between extension lines.
- Off: draws text outside if there is insufficient space between extension lines.

Dim line inside

Specifies whether the dimension line is forced between extension lines:

- **On**: forces dimension line always between extension lines.
- **Off**: draws dimension line outside if there is insufficient space between extension.

Text movement

Specifies what happens when text is moved away from its default location:

- Keep dim line with text: moves the dimension line with the text, stretching the extension lines.
- Move text, add leader: draws a leader between the text and the dimension line.
- Move text, no leader: does not draw a leader.

Dim scale overall

Specifies the overall scale factor for dimensions. This affects the size of arrows and text only.

This setting is not editable for annotative dimension styles. It is recommended to set Dim Scale Overall = 1 for annotative dimension styles.

Place text manually

Toggles whether the user must always specify the text location when creating dimensions.

Dim line forced

Forces the dimension line to be always drawn; forces leaders to be drawn with the DIMDIAMETER and DIMRADIUS commands.

Primary units

Primary units	
Dim units	Decimal
Dim precision	0.00
Fractional type	Horizontal
Decimal separator	,
Dim round	0
Dim prefix	
Dim suffix	
Dim sub-units suffix	
Dim scale linear	1
Dim sub-units scale	100
Suppress leading zeros	☐ Suppress leading zeros
Suppress trailing zeros	☑ Suppress trailing zeros
Suppress zero feet	☑ Suppress zero feet
Suppress zero inches	☑ Suppress zero inches
Dim angle units	Decimal degrees
Dim angle precision	0
Suppress angle leading zeros	☐ Suppress trailing zeros
Suppress angle trailing zeros	☐ Suppress trailing zeros

Dim units

Specifies the display units for dimensions:

- Scientific: scientific notation, such as 1.2345E+01
- Decimal: metric, such as 1.2345
- Engineering: feet and decimal inches, such as 1'-2.3456"
- Architectural: feet and fractional inches, such as 1'-2 1/16"
- Fractional: fractional inches only; no feet, such as 14 1/16"
- Windows Desktop: uses the units set by Windows

Dim precision

Specifies the precision of units, either decimal places or fractional accuracy.

Fractional type

Specifies how fractions are stacked:

- Horizontal: stacks fractions vertically, with a horizontal separator line.
- Diagonal: stacks diagonally, with a diagonal separator line.
- None: does not stack fractions, placed horizontally with a slash separator.

Decimal separator

Specifies the character used to indicate the decimal point; can be any character.
North American countries use the period; European countries use the comma.

Dim round

Specifies the rounding of decimal numbers. The range goes from none to 8 decimal places.

Dim prefix
Specifies a prefix text that appears in front of the dimension text, if any.

Dim suffix
Specifies a suffix text that appears after the dimension text, if any.

Dim sub-units suffix
Specifies suffix text in case the subunit is used. For example, enter 'cm' to display 0.96 as 96cm.

Note: suppress leading zeros must be set to Yes to display dimension distances less than one unit in sub units.

Dim scale linear
Specifies the scale factor for linear dimension values, such as 25.4 for changing inches to millimeters. Positive values apply to dimensions in both model space and paper space. Negative values apply to paper space dimensions only.

Dim sub-units scale
Sets the scale factor for the sub-units. For example, enter 10 if the drawing unit is cm, and the subunit suffix is mm.

Note: suppress leading zeros must be set to Yes to display dimension distances less than one unit in subunits.

Suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zeros, such as 0.23

Suppress trailing zeros
Toggles the display of zeros after the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Suppress zero feet
Toggles the display of zero feet. For example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Suppress zero inches
Toggles the display of zero inches; for example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Dim angle units
Specifies the format of units in angular dimensions:

- Decimal Degrees: 360 degrees in a circle. For example, 123.45 degrees
- Deg/Min/Sec: degrees, minutes, seconds. For example, 123d 12' 45.67"
- Grads: 400 grads in a circle. For example, 230g
- Radians: 2pi radians in a circle. For example, 1.5r

Dim angle precision

Specifies the number of decimal places. The range goes from 0 to 8.

Suppress angle leading zeros

Toggles the display of zero degrees. For example, 0.1234 degrees is displayed as:

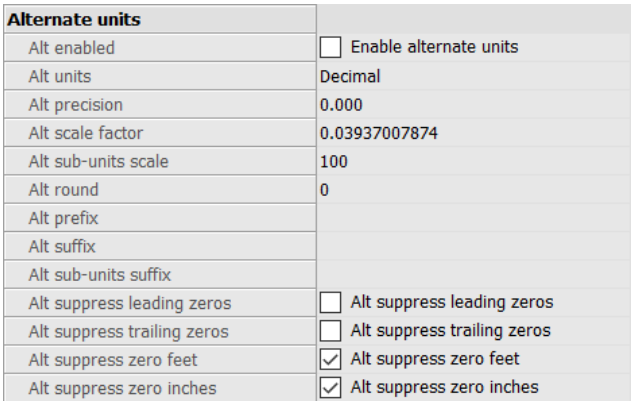
- On: suppresses zero degrees, such as .1234
- Off: allows zero degrees, such as 0.1234

Suppress angle trailing zeros

Toggles the display of zeros after degrees. For example, 0.1200 degrees is displayed as:

- On: suppresses trailing zeros, such as 0.12
- Off: allows trailing zeros, such as 0.1200

Alternate units



Alt enabled

Toggles the display of alternate units:

- On: second, alternative units displayed to the right of primary units.
- Off: only primary units are displayed.

Alt units

Specifies the alternate unit type for linear dimensions:

- Scientific: scientific notation, such as 1.2345E+01.
- Decimal: metric, such as 1.2345
- Engineering: feet and decimal inches, such as 1'-2.3456"
- Architectural Stacked: such as 4'-6.61"
- Fractional Stacked: such as 54 1/2
- Architectural: feet and fractional inches, such as 1'-2 1/16"
- Fractional: fractional inches only; no feet, such as 14 1/16"
- Windows Desktop: uses the units set by Windows

Alt precision

Specifies the precision of alternate units, either decimal places or fractional accuracy.

Alt scale factor
Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).

Alt sub-units scale
Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).

Alt round
Specifies the rounding of alternate numbers. The range goes from zero to 8 decimal places.

Alt prefix
Specifies a prefix text that appears in front of the alternate text, if any.

Alt suffix
Specifies a suffix text that appears after the alternate text, if any.

Alt sub-units suffix
Specifies suffix text in case the subunit is used.

Note: suppress leading zeros must be set to Yes to display alternate distances less than one unit in subunits.

Alt suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Alt suppress trailing zeros
Toggles the display of zeros in front of the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Alt suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Alt suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Tolerances

Tolerances	
Tolerance display	☐ Display tolerance
Limits display	☐ Generate dimension limits as defa...
Tolerance precision	0.00
Tolerance limit lower	0
Tolerance limit upper	0
Tolerance text height	1 mm
Tolerance position vertical	Bottom
Tolerance suppress leading zeros	☐ Tolerance suppress leading zeros
Tolerance suppress trailing zeros	☑ Tolerance suppress trailing zeros
Tolerance suppress zero feet	☑ Tolerance suppress zero feet
Tolerance suppress zero inches	☑ Tolerance suppress zero inches
Alt tolerance precision	0.000
Alt tolerance suppress leading zeros	☐ Alt tolerance suppress leading zeros
Alt tolerance suppress trailing zeros	0
Alt tolerance suppress zero feet	☑ Alt tolerance suppress zero feet
Alt tolerance suppress zero inches	☑ Alt tolerance suppress zero inches

Tolerance display
Toggles the display of tolerance text.

Limits display
Sets the default text to dimension limits.

Tolerance precision
Specifies the display precision of tolerance text. The range goes from 0 to 8 decimal places or 1/1 to 1/256 inch.

Tolerance limit lower
Specifies the value of the lower tolerance.

Tolerance limit upper
Specifies the value of the upper tolerance.

Tolerance text height
Specifies the height of the tolerance text.

Tolerance position vertical
Locates the tolerance text relative to the dimension text:

- Bottom: aligns tolerance text with the bottom of the dimension text.
- Middle: aligns tolerance text with the middle of the dimension text.
- Top: aligns tolerance text with the top of the dimension text.

Tolerance suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Tolerance suppress trailing zeros
Toggles the display of zeros behind the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Tolerance suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Tolerance suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Alt tolerance precision
Specifies the number of decimal places for tolerances in alternate units.

Alt tolerance suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Alt tolerance suppress trailing zeros
Toggles the display of zeros behind the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Alt tolerance suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Alt tolerance suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

9.74 -DIMSTYLE command

Creates and modifies dimensions styles at the Command line.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: -DST

9.74.1 Method

Run the command to create a new dimension style by choosing one of the options. The command also prompts the name of the current dimension style.

9.74.2 Options within the command

?
Lists the names of all dimension styles loaded into the current drawing, other than Standard.

ANnotative
Sets the annotative property of the dimension style.

Apply
Changes or restores the style of existing dimensions by applying the current style.

Save
Saves the current dimension settings as a style.

Note: This option is useful for saving overrides as a style.

STatus
Displays the current value of every dimension variable.

Variables
Lists the values of dimension variables belonging to the selected dimension.

Restore
Resets the values of dimension variables to those of the selected dimension.

9.75 DIMSTYLESET command

Reports the current dimension style in the Command line.

Shape Lite Pro Mechanical BIM

9.75.1 Method

The name of the current dimension style is also shown in the BricsCAD application window as Current Dimension Style in the Status bar.

9.76 DIMTEDIT command

Changes the position of dimension text.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMTED

9.76.1 Description

Allows you to change the position and angle of the text within a dimension entity.





9.76.2 Method

This command offers four methods to change the position of dimension text.

- Rotate the dimension text.
- Move the dimension text to the left.
- Move the dimension text to the right.
- Center the dimension text.

9.76.3 Options within the command

Angle
Rotates the dimension text from the original dimension text position (1) to the rotated dimension text position (2).

Left
Moves the dimension text next to the left extension line of the dimension, depending on the original positioning of the dimension.

Center
Centers the dimension text between the two extension lines of the dimension.

Right
Moves the dimension text next to the right extension line of the dimension, depending on the original positioning of the dimension.

REstore
Resets the rotation angle of the dimension text to 0 degrees and does not change repositioned dimension text.

9.77 DISH command

This command is obsolete and exists for backward compatibility only. Use the AI_DISH command instead.

Shape Lite Pro Mechanical BIM

Icon:

9.78 DISSOLVESKETCHFEATURE command

Dissolves some or all sketch features but keeps the corresponding 3D solid.

Shape Lite Pro Mechanical BIM

9.78.1 Method

Specify a comma separated list with Sketch Based Features (for example, Extrude_1, Loft_1) to be dissolved. Otherwise, the command will dissolve all existing Sketch Based Features by removing their sketch features if there is more than one.

Note: The corresponding 3D solids will remain.

9.79 DIST command

Reports the distance and angle between two points.

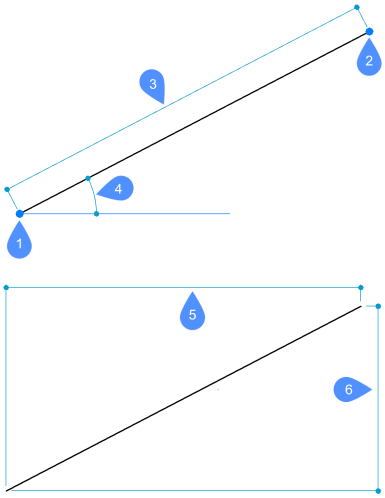
Shape Lite Pro Mechanical BIM

Icon:

Alias: DI

9.79.1 Method

The distance can be measured between two points or multiple points.



1 Starting point for distance

2 End point

3 Distance

4 Angle in XY plane

5 X Distance

6 Y Distance

Note: When DRAGSNAP = ON and the Dynamic Dimensions are enabled, the distance is displayed in the dynamic entry field when hovering over the second point using an appropriate entity snap.

To improve the workflow, the command allows you to use as the starting point the last point previously picked in the program. To do so, press **Enter** after launching the command (when prompted for the starting point).

Note: The last picked point is specified by the LASTPOINT system variable.

9.79.2 Options within the command

Multiple points
Allows you to measure the total distance between multiple points.

Arc
Allows you to specify an arc distance through different options.

Angle
Specify the angle for the arc.

CEnter
Specify the center of the arc. Hold the Ctrl key to switch direction.

Direction
Specify the direction of the arc.

Line
Allows you to specify a line distance.

Radius
Specify the radius of the arc.

Second point
Allows you to specify the arc's second point before specifying its end point.

CLose
Closes the command.

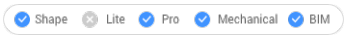
Length
Allows you to add a distance.

Undo
Removes the last point.

Total
Prints the total length in the Command line and concludes the command.

9.80 DISTANTLIGHT command

Places distant lights for renderings.



Icon:

9.80.1 Description

Places distant lights for use with renderings. Distant lights represent sources that are far off, such as the sun, and so does not display a glyph in drawings.

Note: having DEFAULTLIGHTING=1 will open a dialog box where you are asked.

9.80.2 Options within the command

Name
Specifies a name for the light.

Intensity factor
Indicates the relative brightness of the light.

Status
Toggles the light on and off

Photometry
Specifies the parameters for color and intensity.

shadoW
Specifies the look of shadows, if any.

filterColor
Sets the color of the light.

Vector
The direction of the light beams can be defined by a vector, whose XYZ tuple determines the direction of the light, starting at infinity. The default direction points up, in the Z direction.
Unlike regular vectors, this lighting vector specifies only direction, not magnitude.



1: Distant light illuminating from default vector direction.

9.81 DIVIDE command

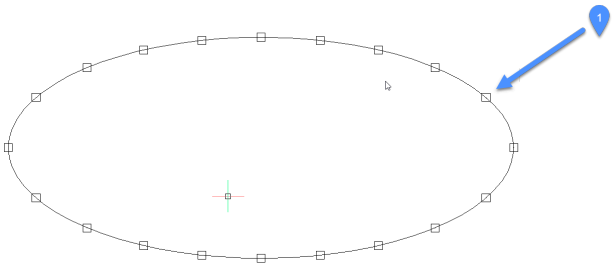
Places a specific number of points or blocks spaced equally along an entity.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIV

9.81.1 Method



1. Points placed evenly divided along entity

The command allows creating and placing points or blocks along the perimeter of an entity, at equal distances.

The following entities can be divided:

- Line
- Polyline
- Spline
- Arc
- Circle
- Ellipse

Note: It might be necessary to adjust the Point display mode settings (DDPTYPE) to display the points correctly.

9.81.2 Options within the command

Number of segments
An integer number between 2 and 32767 can be specified. This will represent the number of segments, the equally-divided distances between points.

Note: Entering 2 for the number of segments places a single point or block at the midpoint of the entity, as the entity is divided into two segments. There is always one more segment than points placed, as no point or block is placed at the start or endpoints of the entity.

insert Block
Place equally-spaced blocks along the entity, instead of points. You can specify the name of the block or use ? to list available blocks.

Yes – align blocks
Rotates blocks for them to match the local orientation of the entity, such as along an arc.

No – do not align
The blocks maintain their orientation (they do not rotate).

9.82 DMANGLE3D command

Creates an angle constraint between faces and/or edges of 3D solids, WCS subentities, and subentities of a coordinate system of a block.

Shape Lite Pro Mechanical BIM

Icons: 

9.82.1 Description

Creates a Planar Angle constraint by default: coordinate planes of the WCS are used as third reference objects whenever possible. It also allows to control the top angle of a cone. Press the TAB key to select obscured geometry.

9.82.2 Options within the command

cone Angle constraint
Allows you to control the top angle of a cone by specifying the angle between the axis and the cone face (= cone half angle). Values must be smaller than 90°.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose the subentity of the coordinate system of the block as an argument. You can choose from: X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane.

Set reference entity
Allows you to manually specify the reference entity:

Keep as is
Allows you to keep the proposed reference object.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose the subentity of the coordinate system of the block as an argument.

None
Creates an Angle constraint instead of a Planar Angle constraint, without a third argument.

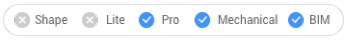
Geometry-driven
Accepts the current value and creates a geometry-driven constraint.

Note: The constraint value displays when hovering over the 3D angle constraint widget.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

9.83 DMAUDIT command

Analyzes and repairs errors.



Icon: 

9.83.1 Description

This command is a powerful tool to analyze and automatically fix problems in 3D geometry supported by ACIS kernel (3D solids, surfaces).

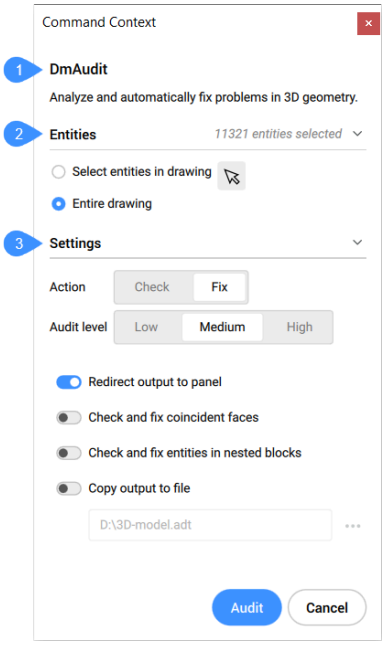
Note: It is recommended to run DMAUDIT whenever 3D geometry is imported.

9.83.2 Method

There are two methods to audit the drawing by selecting one or more entities or the entire model:

- Check problems
- Fix problems

The command opens the **Command Context** panel.



1 Description

2 Entities

3 Settings

Entities:

- **Select entities in the drawing**: by default, the panel lets you choose which entities to select.
- **Entire drawing**: selects all entities in the model space.

Settings:

Action

Lets you set the audit mode.

- **Check**: checks the 3D geometry in the drawing for errors without fixing them.
- **Fix**: by default, repairs errors.

Audit level
Sets the strictness of the check.

- **Low**: runs basic analysis (for example, catching common fatal topology errors).
- **Medium**: this is the default level.
- **High**: goes deep (for example, catching self-intersections in solid bodies).

Redirect output to panel
Toggled on by default to redirect the output of the command to the **Report** panel.

Note: Depending on the value of REPORTPANELMODE variable, it might be necessary to click on the red exclamation mark in the lower right corner to see more details in the **Report** panel.

Check and fix coincident faces
Toggle on to check and fix coincident faces. Requires audit level set to High.

Check and fix entities in nested blocks
Toggle on to check and fix entities in nested blocks.

Copy output to file
Toggle on to report the output of the command in a file.

Note: The options within the command line reflect the options within the **Command Context** panel.

9.84 DMAUDITALL command

Analyzes and repairs errors in drawings inserted as external references

Shape Lite Pro Mechanical BIM

9.84.1 Description

It is a powerful tool to analyze and automatically fix problems in 3D geometry supported by ACIS kernel (3D solids, surfaces).

Note: It is recommended to run DMAUDITALL whenever 3D geometry is imported.

9.84.2 Method

There are two methods to audit the drawing:

- Check problems.
- Fix problems.

9.84.3 Options within the command

Check
Checks the 3D geometry in the drawing for errors without fixing them.

Fix
Repairs errors.

Back
Returns to the main prompt of the command.

report Panel
Redirects the output of the command to the Report panel.

Note: Depending on the value of REPORTPANELMODE variable, it might be necessary to click on the red exclamation mark in the lower right corner to see more details in the **Report** panel.

copy to File
Reports the output of the command in a file.

audit Level
Sets the strictness of the check.

Low
Runs basic analysis (for example, catching common fatal topology errors).

High
Goes deep (for example, catching self-intersections in solid bodies).

Medium
This is the default level.

Coincident faces
Checks and fixes coincident faces. Requires audit level set to High.

9.85 DMBEND command

Bends a 3D solid around an axis.

Shape Lite Pro Mechanical BIM

Icon: 

9.85.1 Method

Select or draw a bend line on the face of the 3D solid and then select the angle and radius of the bend using the mouse or by entering the angle and radius values.

Note: If the solid was created as a Sketch Based Feature solid (CREATESKETCHFEATURE system variable is ON), a warning prompts you to confirm if you want to remove the feature.

9.85.2 Options within the command

Select a line to be used as a bend axis
Allows you to select an existing line.

New line
Creates a new line to be used as a bend axis.

Left
Swaps the fixed side to the left.

Middle
Leaves both sides mobile.

Right
Swaps the fixed side to the right.

Note: The Hotkey Assistant widget displays if the HOTKEYASSISTANT (HKA) system variable is ON. Repeatedly press the **Ctrl** key during the dynamic display of the command to select an option for the fixed side of the bend, **Left**, **Middle**, or **Right**.



Settings
Sets the values for angle and radius.

Angle
Sets the bend angle.

Radius
Sets the bend radius.

Accept model
Accepts the model.

9.86 DMCHAMFER command

Creates equal and variable distance chamfers between adjacent faces sharing a sharp edge.

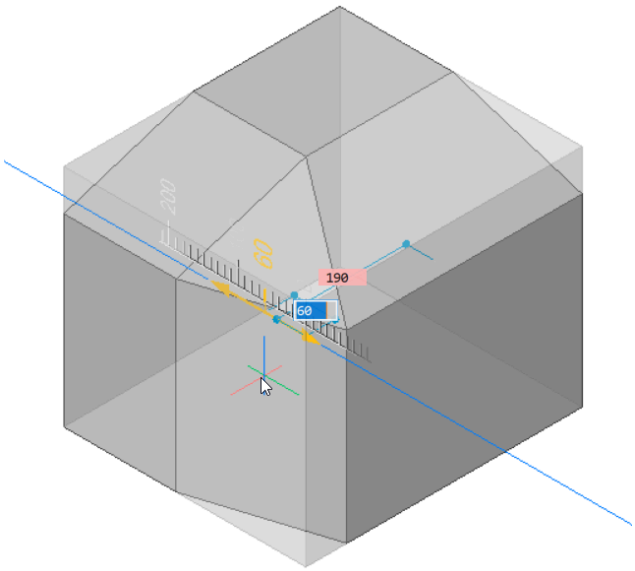
Shape Lite Pro Mechanical BIM

Icons:

9.86.1 Description

After selecting the edges to create chamfer you may choose to specify a constant symmetrical offset or one of the available advanced options.

The chamfer is applied dynamically, and the manipulator is displayed.



Note: When the CREATESKETCHFEATURE system variable is ON and only for default DMCHAMFER mode, chamfer features are created and are visible in the **Mechanical Browser** panel. The **Distance** property can be modified, but only valid values will be accepted.

9.86.2 Options within the command

Allows to create variable chamfers. All advanced mode chamfer procedures start by creating a symmetrical offset, which is then modified by specifying offsets and/or angles. Before specifying the final value, you can press the TAB key to modify previously specified values.

ASymmetrical
Allows you to specify two offsets.

ANgular
Allows you to specify an offset and an angle.

Variable Symmetrical
Allows you to specify two offsets.

VAriable aSymmetrical *
Allows you to specify four offsets.

VAriable aNgular *
Allows you to specify two pairs of offsets and an angle.

* Creates a curved chamfer face.

9.87 DMCOINCIDENT3D command

Applies a coincident constraint between two subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icons: 

9.87.1 Method

Applies a coincident constraint between two edges, two faces, an edge and a face or a vertex and a face or edge of two different solids or surfaces.

9.87.2 Options within the command

Autoconstrain
Automatically creates a coincident constraint between each pair of connected faces in selection of solids. You can check which constraints are created in the **Mechanical Browser** panel or in the **Parameters Manager** panel.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.88 DMCONCENTRIC3D command

Applies a concentric constraint between two circular subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.88.1 Method

The concentric constraint could be applied between any two circular surfaces or circular edges of a 3D entities (cylindrical (circular or elliptical), spherical or conical surfaces and their edges) or between a subentity of a 3D entity and a 2D circular entity.

Select the two circular subentities between which the concentric constraint will be applied. Press the TAB key to select obscured geometry.

9.88.2 Options within the command

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.89 DMCONSTRAINT3D command

Applies geometric and dimensional constraints for 3D entities.

Shape Lite Pro Mechanical BIM

9.89.1 Description

Applies geometric and dimensional constraints between faces, surfaces or edges of 3D entities.

The geometric constraints create a specific relationship between two subentities/entities, like parallelism, perpendicularity, tangency, coincidence, concentricity, fixed or rigid.

The dimensional constraints restrain the value of a dimension of an entity or between entities, like radius, angle or distance. It is also possible to specify the lower or/and upper bounds for the constraint value, and remove them if desired.

Select the subentities between which you will apply a dimensional or geometric constraint.

Press the Tab key to select obscured geometry.

9.89.2 Options within the command

New
Allows to create a new parameter.

Fix
Applies a fixed constraint to a solid or to an edge or a face of a solid. The option is equivalent to the DMFIX3D command.

Multiple
Selects multiple entities.

BLock
Selects a block reference.

Coincident
Applies a coincident constraint between two edges, two faces, or an edge and a face of two different solids. The option is equivalent to the DMCOINCIDENT3D command.

Autoconstrain
Creates automatic constraints.

WOrld
Select a reference coordinate system subentity.

PAth
Allows a point to move along a curve by specifying the curve's parameters with an expression. The option is equivalent to the DMPATH3D command.

CONcentric
Applies a concentric constraint between two cylindrical, spherical or conical surfaces. The option is equivalent to the DMCONCENTRIC3D command.

Parallel
Applies a parallel constraint between two faces of a solid or of different solids. The option is equivalent to the DMPARALLEL3D command.

PErpendicular
Applies a perpendicular constraint between two faces of a solid or of different solids. The option is equivalent to the DMPERPENDICULAR3D command.

Tangent
Applies a tangent constraint between a face and a curved surface of different solids. The option is equivalent to the DMTANGENT3D command.

RIgidset
Allows to define a set of entities or subentities as a rigid body. The option is equivalent to the DMRIGIDSET3D command.

Distance
Applies a distance constraint between two subentities of a solid or of different solids. The option is equivalent to the DMDISTANCE3D command.

Radius
Applies a radius constraint to cylindrical surfaces or circular edges. The option is equivalent to the DMRADIUS3D command.

Angle
Applies an angle constraint between two faces of a solid or of different solids. The option is equivalent to the DMANGLE3D command.

Edit
Allows to edit the named constraint

NodeId
Specify the constraint by its ordered number.

Enable
Enables/disables the selected constraint.

change Arguments
Allows to change the 3D constraint arguments.

Replace argument
Specify the index of the argument to be replaced and its replacement.

Exclude argument
Specify the index of the argument to be excluded.

Note: This option is available only for RigidSet constraints.

Add arguments
Specify the position where the new arguments to be added.

Note: This option is available only for RigidSet constraints.

Append arguments
The new argument is added at the end of the list.

Edit Arguments
Allows to edit the list of 3D constraint arguments by selecting entities or by choosing one of the options.

Note: This option is available only for RigidSet constraints.

Edit options specific for dimensional constraints:

Value
Specifies a new value for the parameter

Lower Bound
Specifies a lower bound for the parameter.

Upper Bound
Specifies an upper bound for the parameter.

eXpose
Specifies whether the dimensional constraint is exposed when the drawing is inserted in another drawing.

Geometry-driven
Specifies whether the parameter is geometry driven.

Properties
Allows you to edit the arguments of the parameter.

Placements
Specifies the placement of the arguments.

Direction
Specifies the new direction of the constraint.

Use as
Specifies how to use the arguments.

Rename
Allows you to rename a constraint.

Delete
Allows you to delete a constraint.

?
Lists the constraints and their status.

9.90 DMCOPYFACES command

Copies a feature from a 3D solid.

Shape Lite Pro Mechanical BIM

Icon:

9.90.1 Description

Copies a set of 3D solid faces forming a protrusion or depression feature, such as holes, pockets, extrusions, ribs, and their combinations within the same 3D solid or from one 3D solid to another.

9.90.2 Options within the command

Select entities
Select all faces of the entire feature.

Base point
Specify the base point.

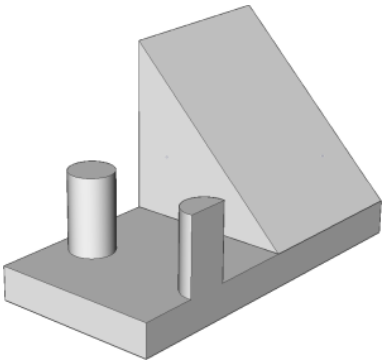
Insertion point
Allows you to select an insertion point.

Rotate
Rotates the faces around the Z axis of the UCS.

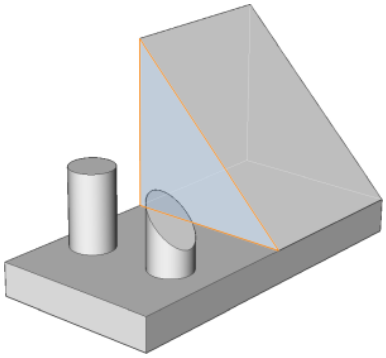
mUltiple
Insert multiple copies of a feature.

MOde
Toggles between various cutting options.

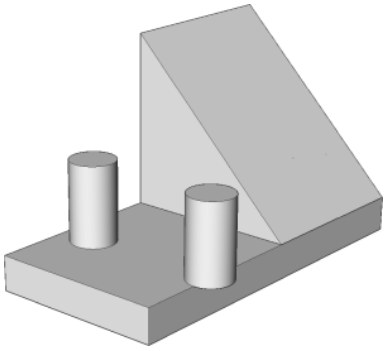
Placement Face
Cuts the copied feature so that it suits the placement face.



Select Faces
Cuts the copied feature so that it suits the selected face.



No Cut
Does not cut the copied feature.



9.91 DMDEFORMCURVE command

Deforms 3D solids or surfaces by replacing their edges with given curves.

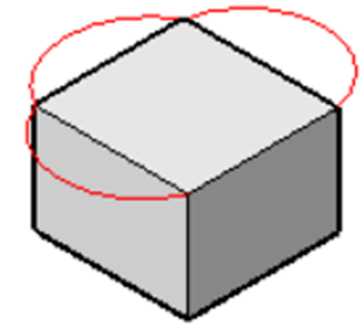
Shape Lite Pro Mechanical BIM

Icon:

9.91.1 Description

Deforms one or more connected faces of a 3D solid or a surface by replacing their edges with given curves.

Select the edges that will be replaced with the target curves to get the new 3d solid or surface.





9.91.2 Options within the command

add Faces
Allows to select additional faces, other than the faces adjacent to selected edges, to deform.

Multiple edges
Allows the deformation of a chain of edges to a single target curve.

9.92 DMDEFORMMOVE command

Deforms 3D solids, surfaces or regions by moving and/or rotating their edges.

Shape Lite Pro Mechanical BIM

Icon:

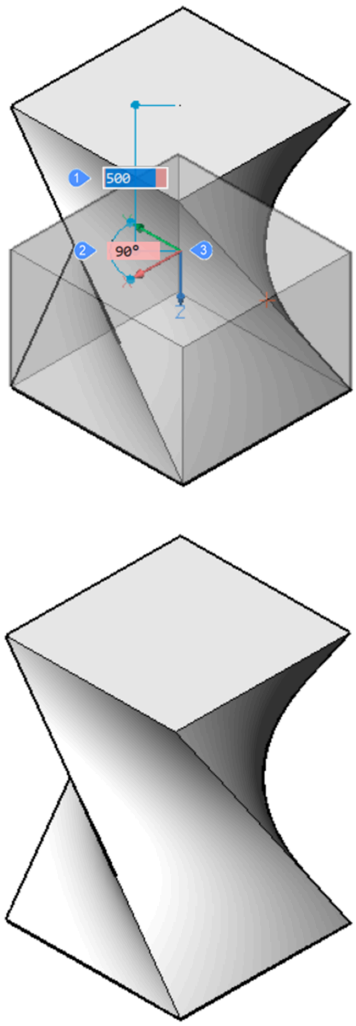
9.92.1 Description

Deforms connected faces of a 3D solid or a surface by moving and/or rotating their edges.

To specify the deformation do one of the following:

- Specify a point in the drawing.
- Type a value in the dynamic entry fields.
- Hit the TAB key to switch between the distance (1) and angle (2) fields.

The edges are rotated around an axis passing through the base point (3) in the deformation direction.



Dynamics (left) and final result (right)

9.92.2 Options within the command

Base point
Allows to specify the base point. The deformation axis passes through the base point.

Direction
Allows to specify the direction of the deformation axis. You may choose to define the direction by specifying two points in the drawing or choose one of the UCS axis.

additional Faces
Allows to select additional faces to be deformed, in addition to ones adjacent to the selected edge(s).

9.93 DMDEFORMPOINT command

Deforms a region, face(s) of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.

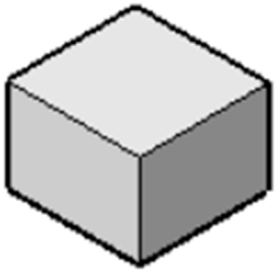
Shape Lite Pro Mechanical BIM

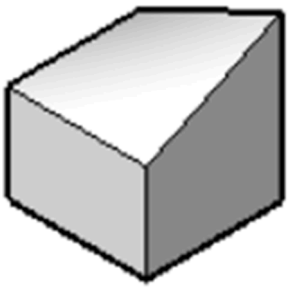
Icon:

9.93.1 Description

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. The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

Select a region or a face or some connected faces of a 3D solid or surface, then specify the deformation point and the deformation value to deform the entity.





9.93.2 Options within the command

change Parameters
Choose a method to change parameters.

Alpha
Specifies the resistance to stretching. It is a second order tensor which can be described with three numbers. Each value must be either 0 or a positive number:

- alpha U: U-direction resistance,
- alpha V: V-direction resistance,
- alpha theta: the angle between U and V principal directions of the surface and the material property directions.

Beta
Specifies the resistance to bending. Similarly to Alpha, Beta is defined as a triple: beta U, beta V, beta theta. Each value must be either 0 or a positive number.

Gamma
Specifies the resistance to the rate of change in bending. The value must be either 0 or a positive number.

Delta
Specifies the resistance to deviations from the default shape. The value must be either 0 or a positive number.

set Target Point
The entities are deformed towards the specified point.

set Direction
The selected entities are deformed dynamically in the specified direction.

change Base point
Select a new deformation point.

9.94 DMDELETE command

Erases entities from the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

The DMDELETE command was integrated into the ERASE command.

9.95 DMDISTANCE3D command

Applies a distance constraint between two entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.95.1 Description

Applies a distance constraint between two subentities of the same solid or of different solids, as well as WCS entities, points, cylinders, and spheres.

Select the two entities/subentities for which the distance constraint will be applied by specifying the distance value. Press the TAB key to select obscured geometry.

9.95.2 Options within the commands

Use as
Gives the option to specify how the distance must be measured between cylindrical and spherical faces. The default measurement mode is Axis.

Boundary
The boundary of the argument.

Axis
An argument that has an axis; circle, cylinder, cone, or torus.

Center
An argument that has a central point; circle, sphere or torus.

Vertex
A cone vertex only.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

Geometry-driven
Accepts the current value and creates a geometry-driven constraint.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

9.96 DMEXTRUDE command

Creates 3D solids or surfaces by extruding entities.

The DMEXTRUDE command has been unified with the EXTRUDE command. Use the EXTRUDE command instead.

9.97 DMFILLET command

Creates a smooth fillet between adjacent faces sharing a sharp edge.

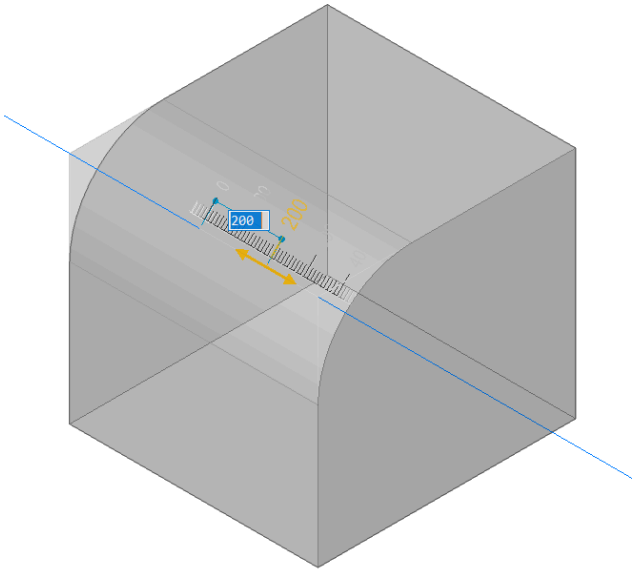
Shape Lite Pro Mechanical BIM

Icon: 

9.97.1 Description

Creates constant or variable radius fillets between adjacent faces that are sharing a sharp edge. After selecting the edges, you can choose to specify a constant radius to fillet edges or one of the available advanced options.

The fillet is applied dynamically and the manipulator is displayed.



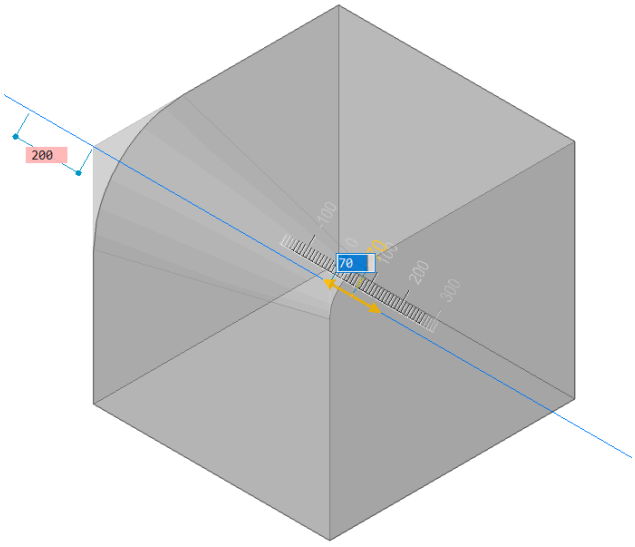
Note: When the CREATESKETCHFEATURE system variable is ON and only for default **Constant radius mode**, fillet features are created and are visible in the **Mechanical Browser** panel. The **Radius** property can be modified, but only valid values will be accepted.

9.97.2 Options within the command

Variable radius mode

Allows you to create a variable fillet between adjacent faces that share (a) sharp edge(s) by specifying the two end radii.

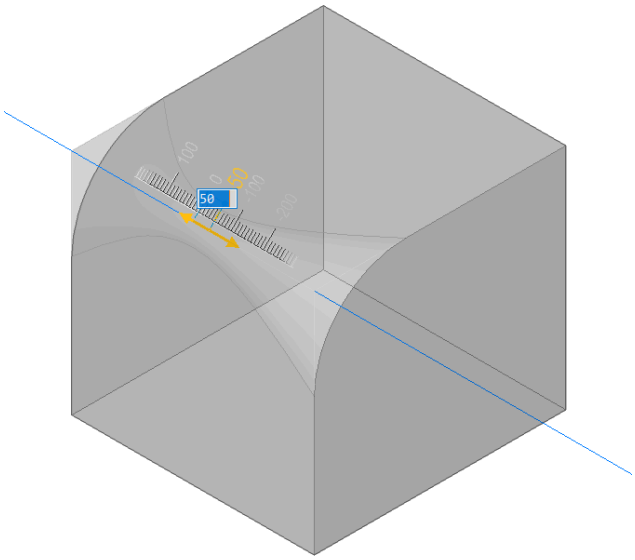
Note: This option is available only for smoothly connected not cyclical edges.



Intermediate radius mode

Allows you to insert intermediate radii by picking points along the selected edges, after specifying a constant radius.

Note: This option is available only for smoothly connected edges.



Erase points
Allows you to erase previously inserted intermediate radii points along the selected edges. Move the cursor on the edge near the desired radius and select the corresponding point to deleted it.

9.98 DMFIX3D command

Applies a fixed constraint to 3D entity.

Shape Lite Pro Mechanical BIM

Icon:

9.98.1 Description

Applies a fixed constraint to a solid or to an edge or a face of a solid.

9.98.2 Options within the commands

Multiple
Allows to select multiple entities or subentities like:

- 3D entities: all faces and edges of the 3D solid or surface will be fixed.
- Faces: the selected faces will be fixed and cannot be moved or rotated.
- Edges: the selected edges will be fixed. The adjacent faces can be rotated, but not moved.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.99 DMGROUP command

Creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

Shape Lite Pro Mechanical BIM

9.99.1 Description

A group feature can be coupled with its description, which can be used to store manufacturing instructions or any other information with your model. Group features usually survive after modification of 3D solids/ surfaces and can be accessed or created in Lisp scripts and BRX applications.

9.99.2 Options within the command

New
Creates a new group.

Edit
Allows to edit an existing group feature by calling its name.
Use Shift to select entities from this selection set.

Dissolve
Allows to delete an existing group feature by calling its name.

Note: Group feature names are case sensitive.

9.100 DMMOVE command

Moves entities or sub-entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.100.1 Description

Moves solids, faces, edges or vertices of a solid or inserts using a vector.

You can select one or more entities. Press Enter to stop selecting. To specify the base and the endpoint you can choose one of the following methods:

- Specify a point by clicking the left mouse button.
- Type a value in the dynamic input field.

The selected geometry moves dynamically.

Note: Geometric and dimensional constraints between selected entities are taken in consideration, as well as the lower and upper bounds parameters for 3D dimensional constraints. Constraints can be accessed and modified through the **Mechanical Browser** panel.

9.101 DMPARALLEL3D command

Applies a parallel constraint between subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.101.1 Method

Select two faces of a solid or two faces of different solids or surfaces or a face and an edge of two different solids between which the parallel constraint will be applied. Press the TAB key to select obscured geometry.

9.101.2 Options within the command

WOrld

Allows you to select the WCS subentity as an argument.

BLock

Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.102 DMPATH3D command

Allows a point to move along a curve by specifying the curve's parameters.

Shape Lite Pro Mechanical BIM

Icon: 

9.102.1 Method

Select any curve entity (polyline, 3D polyline, spline or helix) as the path and a point/vertex from a 3D solid and specify the value of the curve parameter. Press the TAB key to select obscured geometry.

9.102.2 Options within the command

WOrld

Allows you to select the WCS subentity as an argument.

BLock

Allows you to choose, as an argument, the subentity of the coordinate system of the block.

Geometry-driven

Accepts the current value and creates a geometry-driven constraint.

9.103 DMPERPENDICULAR3D command

Applies a perpendicular constraint between two subentities of two 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.103.1 Description

Applies a perpendicular constraint between faces or edges of a solid, or of different solids or surfaces.

Select the two subentities of the 3D entities between which the perpendicular constraint will be applied. Press the TAB key to select obscured geometry.

9.103.2 Options within the command

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.104 DMPUSHPULL command

Adds or removes volume from a solid by moving highlighted faces.

Shape Lite Pro Mechanical BIM

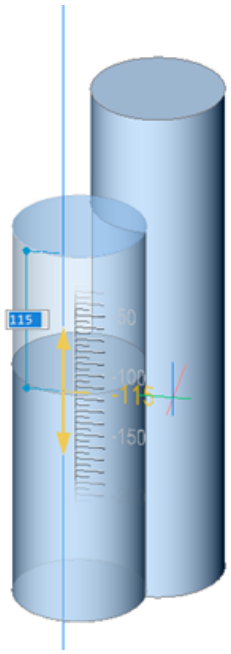
Icon:

9.104.1 Description

Adds volumes to or removes volume from a solid using the cursor movement or direct distance input. You can use this command to edit any face of a 3D solid or a surface. You can select more faces or detected boundaries. Press Enter to complete the selection set.

Note: Plies of multiply compositions are selectable if the Select Faces of the SELECTIONMODES system variable is set.

The selected geometry moves dynamically. Using the Manipulator you can do one of the following:



- You can press the TAB key to select a different reference face. All faces which are parallel to the face being push/pulled are recognized, beginning with the first ones. Press and hold the SHIFT key when pressing the TAB key to cycle reference faces in reverse order. To select a reference face on a different solid, move the cursor over that face, then hit the TAB key. The selected face is highlighted.
- Click a point in the model.

- Type a value in the dynamic distance field.
- Move the cursor to adjust the value in the dynamic distance field. You can optionally zoom in/out to set the snap increment size of the ruler.

9.104.2 Options within the command

Enable subtract
Solids being push/pulled are subtracted from other 3D solids they intersect.

Disable subtract
Solids being push/pulled are not subtracted from other 3D solids they intersect.

Note: When the system variable DMPUSHPULLSUBTRACT = 1, the solid being pushpulled is subtracted from interfering solids. Pressing the Ctrl key allows you to override the DMPUSHPULLSUBTRACT system variable.

Note: If the Hotkey Assistant (HKA) is ON the Hotkey Assistant widget displays, indicating whether the command is in subtract mode or not.

9.105 DMRADIUS3D command

Applies a radius constraint to circular 3d entities or subentities.

Shape Lite Pro Mechanical BIM

Icon: 

9.105.1 Description

Applies a radius constraint to circular edges, cylindrical surfaces, spheres and tori by indicating the radius value or using geometry-driven.

9.105.2 Method

Select a cylindrical or sphere surface or a circular edge and specify the radius value or keep the geometry-driven. In case of selecting a torus surface, you can constrain the rotation radius or/and the tube radius.

Press the TAB key to select obscured geometry.

9.105.3 Options within the command

Rotation radius
Specify the rotation radius to be constrained.

Tube radius
Specify the tube radius to be constrained.

Geometry-driven
Accept the current value and create a geometry-driven constraint.

Note: The constraint value displays when hovering over the 3D radius constraint widget.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

9.106 DMREPAIR command

Analyzes and repairs errors.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The DMREPAIR command has been replaced by the DMAUDIT command.

9.107 DMREVOLVE command

Creates 3D solids or surfaces by revolving 2D entities about an axis.

Shape Lite Pro Mechanical BIM

Icons: 

9.107.1 Description

Note: The *DMREVOLVE* command has been unified with the *REVOLVE* command. Use the *REVOLVE* command instead.

9.108 DMRIGIDSET3D command

Allows to define a set of entities or subentities as a rigid body.

Shape Lite Pro Mechanical BIM

Icon: 

9.108.1 Method

Select a set of edges, faces or 3D entities that will be linked as a rigid body.

All members of a rigid set are moved (DMMOVE command) and rotated (DMROTATE command) together, while their relative positions within the rigid set are preserved.

Press the TAB key to select obscured geometry.

9.108.2 Options within the command

WOrld

Allows you to select the WCS subentity as an argument.

BLock

Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.109 DMROTATE command

Rotates the selected geometry around an axis.

This command is deprecated. Use the ROTATE3D command instead.

9.110 DMSELECT command

Selects edges and faces of 3D solids or surfaces based on their geometric properties.

Shape Lite Pro Mechanical BIM

9.110.1 Description

Allows you to select a group of faces which form a geometric feature, such as a protrusion, a depression, etc. You can combine different options of this command, e.g. select all fillets whose radius is less than a given value. Select the faces or edges whose characteristics will be considered as the selection set's pattern.

9.110.2 Options within the command

Attribute

Radius
Selects faces/edges by radius.

Area
Selects faces by area.

Length
Selects edges by length.

Type
Selects faces/edges by their geometric type.

Offset
Selects faces/edges by their offset value.

Exclude short fillets
Excludes short fillets from the selection. This option might be useful when working with imported sheet metal parts.

Then choose between:

from Sample
Uses the value of the sample face/edge.

Value
Type a value in the Command line.

Note: In case of a non-zero value, you may filter entities as **Less**/**Less or eQual**/**Equal**/**Greater or eQual**/**Greater**.

Relation

Coincident
Selects coincident faces/edges.

Parallel
Selects parallel faces/edges.

coAxial
Selects coaxial faces/edges.

Smooth
Selects the smooth side of a part.

Primitive

Face
Selects faces.

Edge
Selects edges.

Loop
Selects edge loops.

Note: The preselection will determine the result. It is necessary to select a set of connected faces. The command will then select all edges, which describe the border of the selection set; internal edges will be skipped. If the result contains multiple loops and you want only one, the preselection should contain a set of connected faces one edge of the desired output loop. If none of the loops of the selection set contains the selected edge, the output will be empty.

edge Network
Finds an adjacent edge which has similar convexity for a given input edge.

feaTure

Protrusion
Selects protrusions.

Depression
Selects depressions.

Fillet
Selects fillets.

fillet Network
Selects fillet networks.

Chamfer
Selects chamfers.

Chamfer Network
Selects chamfer networks.

Select seed
The selected set of faces /edges should be considered as seed elements for features selection.

Subset
Select the subsets to be considered as seed elements for features selection.

sKip
The entire model is considered as seed for features selection.

Finish
Finishes the command.

9.111 DMSELECTEDGES command

Selects the edges of faces and solids.

Shape Lite Pro Mechanical BIM

Icon:

9.111.1 Method

Select the faces or solids from which all edges will be in the selection set.

Press and hold the CTRL key, then click the edges to be removed from the selection set.

9.112 DMSIMPLIFY command

Simplifies the geometry and topology of 3D solid entities.

Shape Lite Pro Mechanical BIM

Icon:

9.112.1 Description

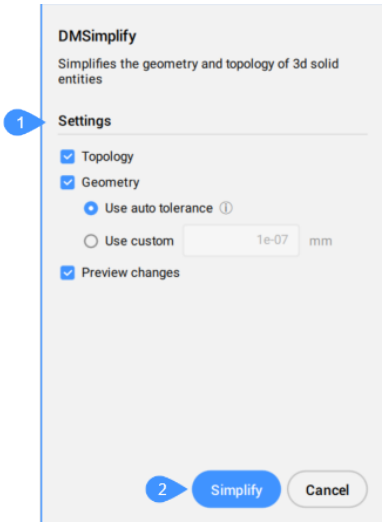
Removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.

Note: It is recommended to always run this command on imported 3D solid geometry.

9.112.2 Method

The **DMSimplify** command context panel is automatically displayed when running the command.

9.112.3 Options within the command context panel



1 Settings

2 Simplify

Settings

Defines what this command will take into consideration.

Topology

If checked, simplifies the topology of the 3D solid entities.

Geometry
If checked, simplifies the geometry of the 3D solid entities.

Use auto tolerance
If checked, the algorithm determines the optimal tolerance based on the solid's geometry.

Note: This is the default option, highly recommended.

Use custom
If checked, it allows you to set a tolerance value.

Preview changes
If checked, displays the **Results** panel.

Note: By default, this option is not checked.



You can choose which detected entities to be simplified. You can select them one by one or all together.

Simplify
When the **Preview changes** button is active, it opens the **Results** panel, otherwise it finishes the command.

Note: After the command ends, a balloon notification with information about the result appears on the bottom right side of the drawing. Pressing the **More details** button opens the **Report** panel for the command.

Note: The options within the command context panel reflect the options within the Command line.

9.113 DMSIMPLIFYALL command

Simplifies the geometry in solids, within drawings inserted as external references.

Shape Lite Pro Mechanical BIM

9.113.1 Description

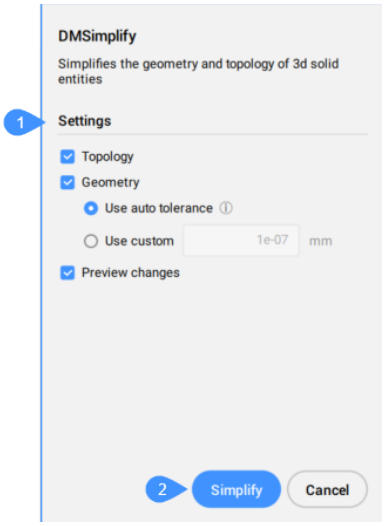
Removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.

Note: It is recommended to always run this command on imported drawings with inserted external references.

9.113.2 Method

When running the DMSIMPLIFYALL command, the **DMSimplify** command context panel is automatically displayed.

9.113.3 Options within the command context panel



1 Settings

2 Simplify

Settings
Defines what this command will take into consideration.

Topology
If checked, simplifies the topology of the 3D solid entities.

Geometry
If checked, simplifies the geometry of the 3D solid entities.

Use auto tolerance
If checked, the algorithm determines the optimal tolerance based on the solid's geometry.

Note: This is the default option, highly recommended.

Use custom
If checked, it allows you to set a tolerance value.

Preview changes
If checked, displays the **Results** panel.

Note: By default, this option is not checked.



You can choose which detected entities to be simplified. You can select them one by one or all together.

Simplify
When the **Preview changes** button is active, it opens the **Results** panel, otherwise it finishes the command.

Note: After the command ends, a balloon notification with information about the result appears on the bottom right side of the drawing. Pressing the **More details** button opens the **Report** panel for the command.

Note: The options within the command context panel reflect the options within the Command line.

9.114 DMSTITCH command

Converts a set of region and surface entities that bound a watertight area to a 3d solid.

Shape Lite Pro Mechanical BIM

Icon:

9.114.1 Method

Select the surfaces to stitch and convert them in a 3D solids or surfaces.

9.114.2 Options within the command

MOde
Allows to set the validation mode either manual or automatic.

Auto
Creates only valid 3d solid or surface entities without free or non-manifold edges.

SOlid
Creates invalid 3d solid entities.

SUrface
Creates invalid surfaces.

Note: Creating an invalid 3d solid entities could be useful for understanding the reason of a failure. The command reports problems (free or non-manifold edges) and highlights them. You can either accept the invalid 3d solid by pressing Enter or refuse it by pressing Cancel.

Tolerance
Type a value or accept **Auto**.

Note: The selected validation mode is stored and re-used for future calls of the command.

9.115 DMSTRETCH command

Stretches one or more 3D solids along an axis.

Shape Lite Pro Mechanical BIM

Icon: 

9.115.1 Description

Allows you to stretch one or more 3D solids along an axis defined by two points. The command creates two intersection surfaces perpendicular to the selected axis and isolates the region of the target solid(s) between those intersection surfaces. You can then stretch this region along the axis direction by moving the mouse or typing a new length for the region.

Note:

- If one of the solids was created as a Sketch Based Feature solid (CREATESKETCHFEATURE system variable is ON), a warning prompts you to confirm if you want to remove the feature.
- If you apply the command to solids containing features, such as holes, doors, or windows, these features retain their original location and size.

9.115.2 Options within the command

set Continuity
Specifies the type of continuity to use (G0 or G1) where the surfaces meet.

Swap fixed side
Swaps the fixed side.

9.116 DMTANGENT3D command

Applies a tangent constraint between circular subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.116.1 Description

Applies a tangent constraint between a face and a curved surface of different 3D entities or a surface and WCS reference coordinate system or a surface and a reference coordinate system of a block reference. Press the TAB key to select obscured geometry.

9.116.2 Options within the command

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose as an argument the subentity of the coordinate system of the block.

9.117 DMTHICKEN command

Creates 3D solids by thickening surfaces, their faces, faces of 3D solids, regions and wire entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.117.1 Description

Creates 3D solids by thickening surfaces, their faces, faces of 3D solids, regions and wire entities by specifying the thickness value.

The following wire entities are accepted: lines, polylines, circles, ellipses, arcs, helices, splines.

Note: If you thicken a multi-face surface of several adjacent faces of one 3D solid/surface, then the corresponding faces remain adjacent on the opposite side of the new 3D solid(s). This differs the DMTHICKEN command from DMEXTRUDE.

Note: When a wire entity is selected, a circle with the radius of the specified thickness value is swept along the wire.

9.117.2 Options within the command

SIngle side
Adds the thickening in the specified direction.

Both sides
Adds thickening in both directions.

9.118 DMTHREAD command

Creates threads on cylindrical faces, which are represented according to drafting standards in drawing views.

Shape Lite Pro Mechanical BIM

Icon: 

9.118.1 Description

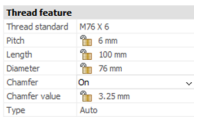
Creates a thread feature in the model that is represented according to drafting standards in drawing views or section view created with VIEWBASE and VIEWSECTION commands.

Note: This command can be entered transparently during commands with 'dmthread.

9.118.2 Method

Selecting a cylindrical face and specifying the pitch and length values a thread feature will be created, that will be represented according to drawing standards in drawing views. With VIEWUPDATE command the changes will be applied in the drawing views.

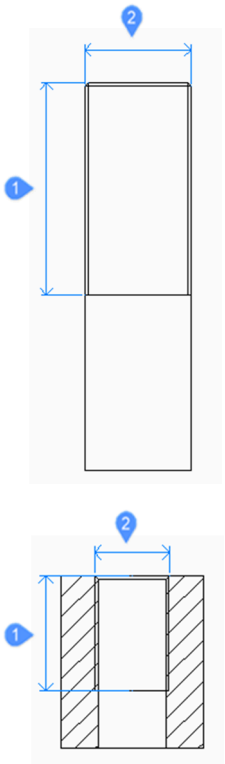
Thread features display in the Mechanical browser , where you can edit the thread parameters.



Note: Male and female thread types are supported.

Note: This command operates only in Model Space.

Below you can see the result of VIEWBASE and VIEWSECTION commands.



1 Length

2 Diameter

9.119 DMTWIST command

Twists a portion of an entity around an axis.

Shape | Lite | Pro | Mechanical | BIM

Icon:

9.119.1 Description

Modifies 3D solids, surfaces or regions by twisting a portion defined by two points around an axis.

Note: You can twist for an angle smaller than 360 degrees.

9.119.2 Method

It is possible to specify the continuity between the deformed and the fixed portion of the entities. You can choose between sharp, smooth and in-between.

9.119.3 Options within the command

Start point of twisting axis
Specify a point where the twisting deformation will start.

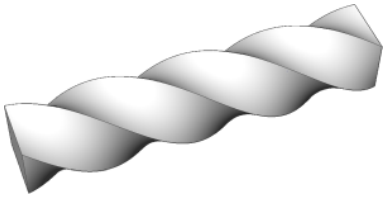
End point of twisting axis
Specify a point where the twisting deformation will end.

Start point of twisting
Specify a point where the twisting will begin.

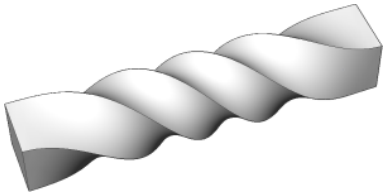
Twist angle
Specify an angle dynamically or type a value.

Continuity
Allows to control whether the transition between the deformed and non-deformed part is sharp or smooth.

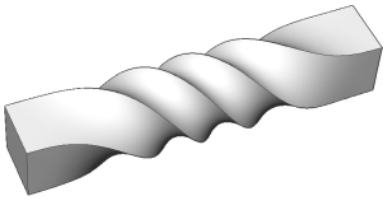
G0
No continuity.



G1
Tangent faces.



G2
Curvature continuity.



9.120 DMUPDATE command

Forces 3d constraints to update.

Shape Lite Pro Mechanical BIM

Icon:

9.120.1 Description

Updates all 3d constraints applied to all 3D entities existing in the drawing.

9.121 DOME command

This command is obsolete and exists for backward compatibility only. Use instead the AI_DOME command.

Shape Lite Pro Mechanical BIM

Icon:

9.122 DONUT command

Creates a closed polyline in the shape of a donut.

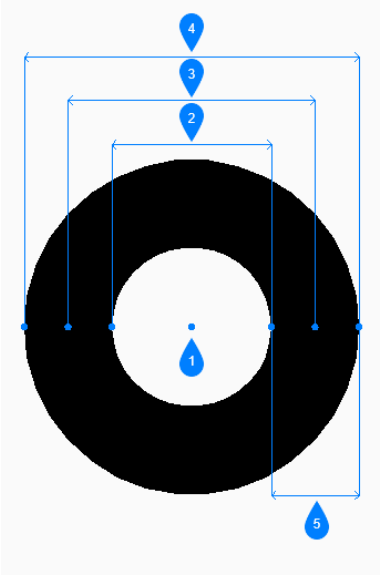
Shape Lite Pro Mechanical BIM

Icon:

Alias: DO, DOUGHNUT

9.122.1 Description

Creates a closed polyline in the shape of a donut from a combination of options including the inside diameter, outside diameter, center and width.



1 Center

2 Inside diameter

3 Diameter

4 Outside diameter

5 Width

9.122.2 Method

This command has 4 methods to begin creating a donut:

- Inside diameter of a donut
- 2 Point
- 3 Point
- Tangent Tangent Radius

9.122.3 Options within the command

Inside diameter of donut
Allows you to begin creating a donut by specifying its inside diameter.

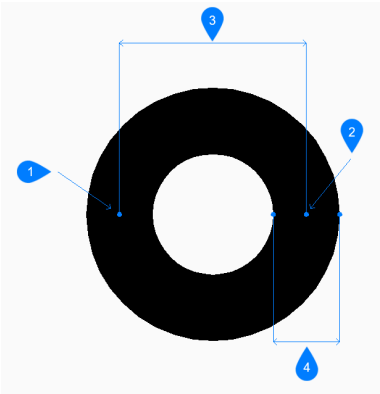
Outside diameter of donut
Specifies the outside diameter of the donut.

Center of donut
Specifies the center of the donut to place it in the drawing.
You can continue placing donuts of the same size until you press **Enter** key to end the command.

2 Point
Allows you to begin creating a donut by specifying its width.

First point on diameter
Specifies a point on the diameter.

Second point on diameter
Specify the second point on the diameter.



1 Point 1

2 Point 2

3 Diameter

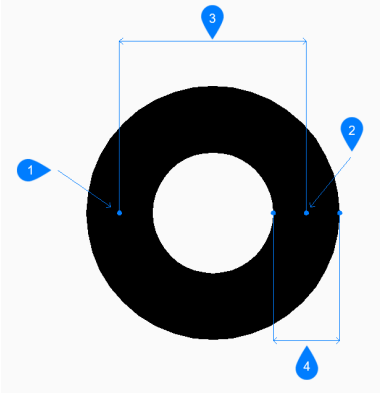
4 Width

3 Point
Allows you to begin creating a donut by specifying its width.

First point on donut
Specifies a point on the circumference of an imaginary circle that runs through the center of its width.

Second point
Specifies the second point on the circumference of the imaginary circle.

Third point
Specifies the third point on the circumference of the imaginary circle.



1 Point 1

2 Point 2

3 Point 3

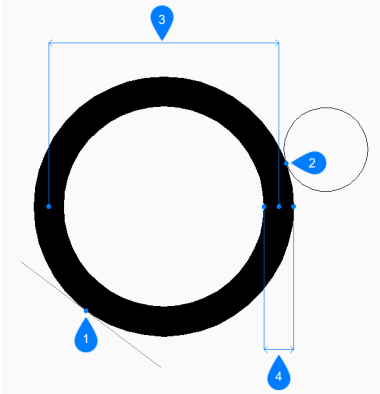
4 Width

Tangent Tangent Radius
Allows you to begin creating a donut by selecting a tangent point on the first entity.

Specify point on object for second tangent on donut
Allows you to select a tangent point on the second entity.

Width of donut
Specifies the width of the donut.

Diameter of donut
Allows you to specify a valid diameter for the donut.
If you specify a diameter that is not possible with the selected tangents, you are prompted to specify the tangents and diameter again.



1 Tangent point 1

2 Tangent point 2

3 Diameter

4 Width

9.123 DRAG command

Moves 3D solids.

Shape Lite Pro Mechanical BIM

Icon:

9.123.1 Description

Moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids.

9.123.2 Method

After selecting one or more planar face(s):

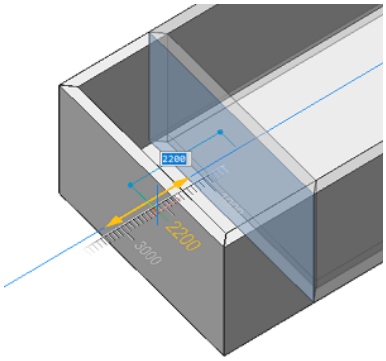
- Type a value in the dynamic distance field.

Note: Toggle Dynamic Input ON from the Status bar.

- Type a value in the Command line.
- Move the cursor to adjust the value in the dynamic distance field.

Note: The selected face(s) move(s) dynamically. The manipulator displays the distance from the current position of the selected face in the dynamic entry field.

Note: Select a different reference face by repeatedly pressing TAB key to cycle all parallel faces of the solid. All parallel faces under the cursor are recognized.



9.123.3 Options within the command

Disable connectivity mode
The connectivity is not kept.

Note: This mode is set as default.

Note: Press Ctrl key to toggle the enable connectivity mode.

Enable connectivity mode
The connectivity is kept.

Note: Press Ctrl key to toggle the disable connectivity mode.

Optionally the Hot Key Assistant can be toggled On, from the status bar, to display the current used connectivity mode.

Note: If the Hot Key Assistant is not displayed, right click on the toggle button and check its configuration.

9.124 DRAGMODE command

Controls the appearance of objects.

Shape Lite Pro Mechanical BIM

9.124.1 Description

Controls the appearance of the objects while being dragged. The command is no longer necessary and is kept only for compatibility.

9.124.2 Options within the command

ON
Turns on DRAGMODE.

Note: Dragged objects are always shown.

OFf
Turns off DRAGMODE.

Note: Dragged objects and the dragline are never shown on the screen.

Auto
Turns on DRAGMODE.

Note: Dragged objects are always shown.

9.125 DRAWINGRECOVERY command

Opens the **Drawing Recovery Manager** panel.

Shape Lite Pro Mechanical BIM

9.125.1 Description

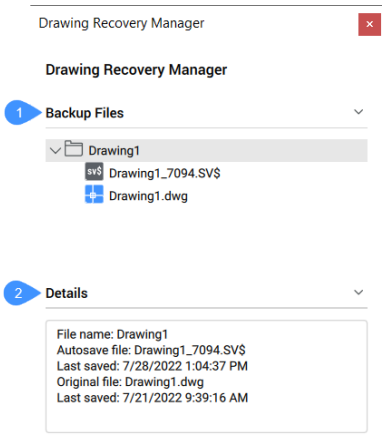
Opens the **Drawing Recovery Manager** panel to display it in the current workspace. The **Drawing Recovery Manager** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Drawing Recovery Manager** panel can be either floating, docked or stacked.

The **Drawing Recovery Manager** panel displays a list of drawing files that can be recovered after a program or system failure.

After a program or system failure, the **Drawing Recovery Manager** panel appears automatically when needed and displays a list of all drawing files that were open, including DWG, DWT and DWS files. A new unsaved drawing can be found in the **Drawing Recovery Manager** panel if the corresponding autosave file (.sv$) was created while editing the new drawing.

Note: You can disable the balloon notification display by toggling off one of the TRAYICONS and TRAYNOTIFY system variables. In this case a drawing recovery message dialog box appears on the Start page instead.

Note: The DRAWINGRECOVERY and DRAWINGRECOVERYHIDE commands can be used to show/hide this panel.



1 Backup Files

2 Details

9.125.2 Backup Files

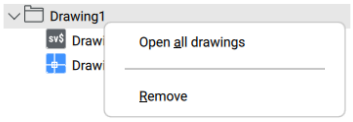
Displays a list of folders containing the drawing files that can be recovered. For each drawing, you choose from the following file types, if they exist:

- DrawingFileName_recover.dwg
- DrawingFileName_a_b_nnnn.sv$
- DrawingFileName.dwg
- DrawingFileName.bak

Recovered drawings are added to the **Recent Files** list and removed from the **Drawing Recovery Manager** panel the next time the program is launched.

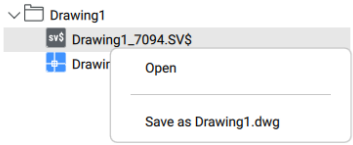
Right-click any item to display the context menu to access different actions depending on the selected item.

Folder:



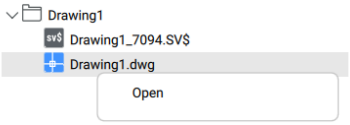
- **Open all drawings**: opens all files from that folder.
- **Remove**: removes the folder from the panel without recovering the files.

Sv$ file:



- **Open**: opens the file.
- **Save as .dwg**: saves the file as a dwg file.

Dwg file:



- **Open**: opens the dwg file.

9.126 DRAWINGRECOVERYHIDE command

Closes the **Drawing Recovery Manager** panel.

Shape Lite Pro Mechanical BIM

9.126.1 Description

Closes the **Drawing Recovery Manager** panel to hide it from the current workspace. If the **Drawing Recovery Manager** panel is stacked when you close it, the **Drawing Recovery Manager** tab or icon is removed from the stack.

9.127 DRAWORDER command

Changes the display order of overlapping entities.

Shape Lite Pro Mechanical BIM

Icon:



Alias: DR

9.127.1 Description

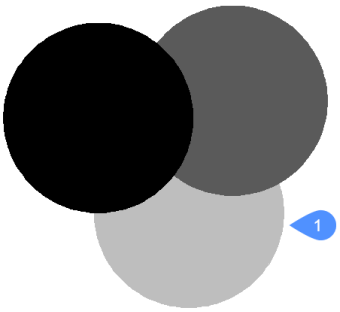
This command is used to display entities over the top of other entities that would otherwise overlap or hide them. This command is meant to work with overlapping objects. While this command works with non-overlapping entities, it has no effect on them.

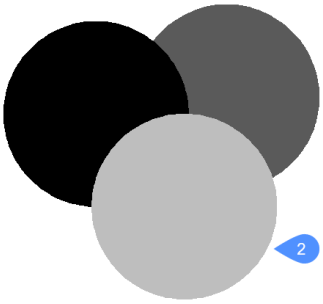
Note: Hatches and text entities in BricsCAD already have handy commands controlling their display order: HatchToBack places all hatching below all other entities, and TextToFront displays all text above all entities.

9.127.2 Method

This command offers 5 methods to change the draw order of overlapping entities:

- Above
- Under
- Bring to front
- Bring to back
- Clear all orders





9.127.3 Options within the command

Above
Moves the visual display of selected entities above the other entities; this option does not necessarily move them to the top (front) of display order.

Under
Moves the visual display of selected entities under the other entities; this option does not necessarily move them to the bottom (back) of the display order.

Clear all orders
Clears the assigned display orders, with entities displayed as they were originally created.

Send to Back
Moves the selected entities' display order below all other overlapping entities (1).

Bring to Front
Moves the selected entities' display order above all other overlapping entities (2).

9.128 DRAWORDERBYLAYER command

Opens the **Open layer list file** dialog box.

Shape Lite Pro Mechanical BIM

9.128.1 Description

Opens the **Open layer list file** dialog box to select a LST file that controls the display order of entities based on their layers.

Note: A LST file is a text file that contains a list of data and it can be created manually or by different programs.

To create a LST file:

1 Open a text editor, such as Notepad, and type the names of the layers you want in your LST file, one layer per line. The names should match the layers of the drawing that need to be modified, in the preferred order.

1 ***Note:*** This determines the order in which the entities are drawn. The first layer written in the list is at the bottom, while the last one is on top.

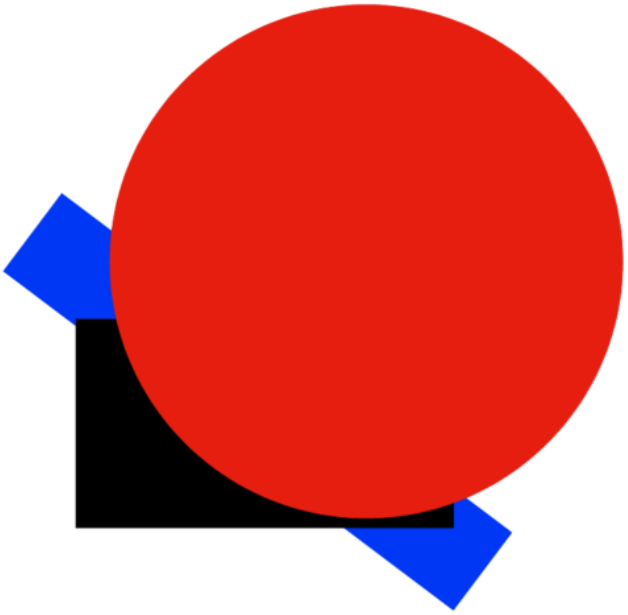
2 Save the text file with the LST extension.

3 Open BricsCAD and load the drawing that need to be modified.

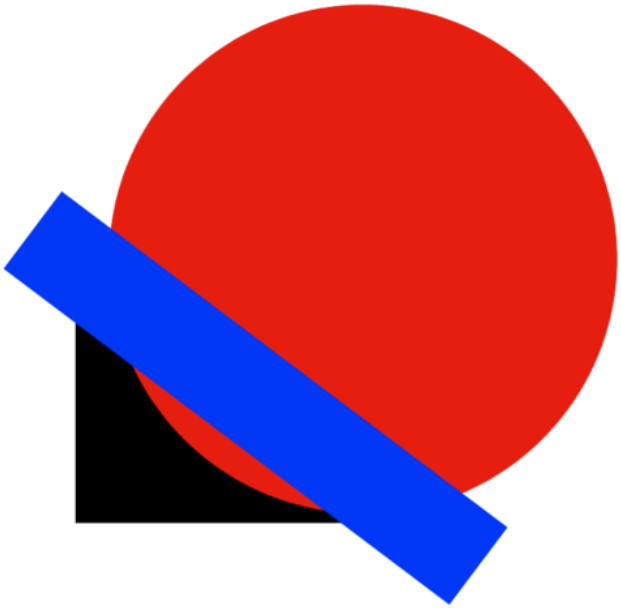
4 Run the DRAWORDERBYLAYER command and select the LST file that you created to apply the changes.

Example:

Draw simple shapes. Each shape should have its own layer.



After using the DRAWORDERBYLAYER command with the file attached at the end of this article, the drawing looks like this:



The LST file used to make changes for this example is: Layer.lst

Note: The layers used in this example are named with the color used for each one of them.

9.129 DSETTINGS command

Opens the **Settings** dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDRMODES, RM

9.129.1 Description

Open the **Settings** dialog box to view and modify system variables. Most, but not all, system variables are available in the **Settings** dialog box. You can modify all system variables using the SETVAR command.

9.130 DTEXT command

Creates a single-line text entity.

Shape Lite Pro Mechanical BIM

Alias: DT

See the TEXT command.

9.131 DUMPSTATE command

Writes information about the current state of the command history

Shape Lite Pro Mechanical BIM

9.131.1 Description

The command is a diagnostics feature that writes some information about the current state of the command history and related internal structures to dumpstate_report.txt file.

9.131.2 Method

Once you encounter a problem with repeating the last command, enter DUMPSTATE at the command line after the incident as soon as possible. The generated file is then placed in the current working directory.

Note: The generated file is overwritten on each DUMPSTATE run.

9.132 DVIEW command

Changes the 3D viewpoint interactively and turns on perspective mode (short for "dynamic view").

Shape Lite Pro Mechanical BIM

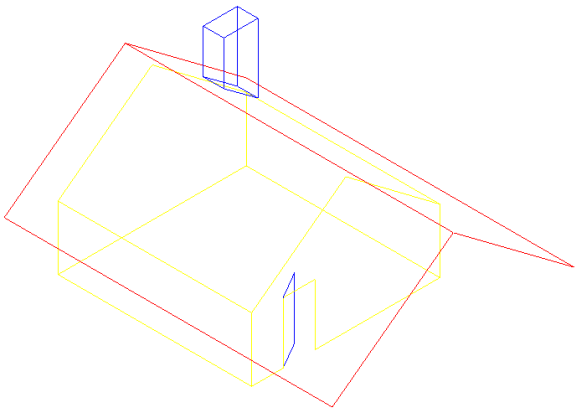
Icon:

Alias: DV

Note: The PERSPECTIVE system variable toggles the perspective property of a view.

9.132.1 Method

Select entities or use DVIEWBLOCK that displays the below built-in 3d block.



9.132.2 Options within the command

rotate CAmera
Specifies the camera (eye) location.

rotate TArget
Specifies the target (look at) location.

Twist view
Sets the twist (view rotation about Z axis) angle.

Distance to target
Specifies the camera to target distance.

Note: This option turns on perspective mode.

x y z POints
Specifies the camera and target location.

CLipping
Sets the clipping distances, which cut off the front and back parts of the model. The front and back clipping planes are positioning perpendicular to the imaginary line between the camera and target.

Note: The clipping planes can be set using the slider bar.

Front clipping
Sets the front clipping plane which obscures objects located between it and the camera.

frONt clipping on
Switch on front clipping at the current clipping distance.

front clipping OFf
Switch off front clipping.

front clipping at Eye
Sets the front clipping plane at the camera location.

Back clipping
Sets the back clipping plane which obscures objects behind it:

back clipping ON
Switch on back clipping at the current clipping distance

back clipping OFf
Switch off back clipping

clipping Off
Removes the clipping planes.

perspective Off
Turns off perspective mode. Use the Distance option to turn perspective mode on.

Hide
Removes hidden lines from the model.

PAn
Interactively pans around the drawing.

Zoom
Zooms in and out interactively (enter a number to change the zoom level).

9.133 DWFOUT command

Saves the drawing in DWF and other formats.

Shape Lite Pro Mechanical BIM

9.133.1 Description

Saves the drawing in DWF and other formats (short for "drawing Web format output"). DWFOUT is an alias for the EXPORT command.

Note: DWF files can be viewed with the DesignReview software, available free from www.autodesk.com/designreview.

9.134 DWGCODEPAGE command

Changes the language code for text in drawings.

Shape Lite Pro Mechanical BIM

9.134.1 Description

Text on computers is defined by the Unicode system, which determines the alphabet used to display text, such as English or Turkish. DWGCODEPAGE changes the code number that specifies the alphabet. This command does not affect the language displayed by the user interface.

9.135 DWGCOMPARE command

Compares a selected drawing with the current drawing.

Shape Lite Pro Mechanical BIM

Icon:

9.135.1 Description

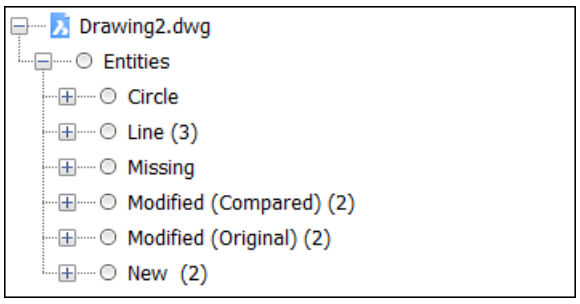
Opens the **Compare** panel to compare a selected drawing with the current drawing, showing the differences.

When the program finds differences, entities are shown in these colors:

- Green (set by the CMPCLRNEW system variable): added entities.
- Red (set by the CMPCLRMISS system variable): removed entities.
- Gray (set by the CMPCLRMOD1 system variable): changed entities in the original drawing.
- Yellow (set by the CMPCLRMOD2 system variable): changed entities in the comparison drawing.

Note: These colour system variables can also be set in the **Settings** dialog box or by clicking on the color preview in the **Compare** panel (under **Legend**).

You can see a list of differences in the **Structure** panel as well. Open it with the STRUCTUREPANEL command and make sure that the Default CST configuration file is loaded. Expand the **Comparison** node in the **Structure** panel.



Note: The ENDCOMPARE command unloads the comparison drawing.

9.135.2 Options within the command

Select

Opens the **Select the file to compare with** dialog box that lets you select a drawing to compare the current drawing to.

Limit

Sets the limit on how many entities to compare in the drawing.

9.136 DWGCOMPARECLOSE command

Closes the **Compare** panel.

Shape Lite Pro Mechanical BIM

9.136.1 Description

Closes the **Compare** panel to hide it from the current workspace. If the **Compare** panel is stacked when you close it, the **Compare** tab or icon is removed from the stack.

9.137 DWGCOMPAREOPEN command

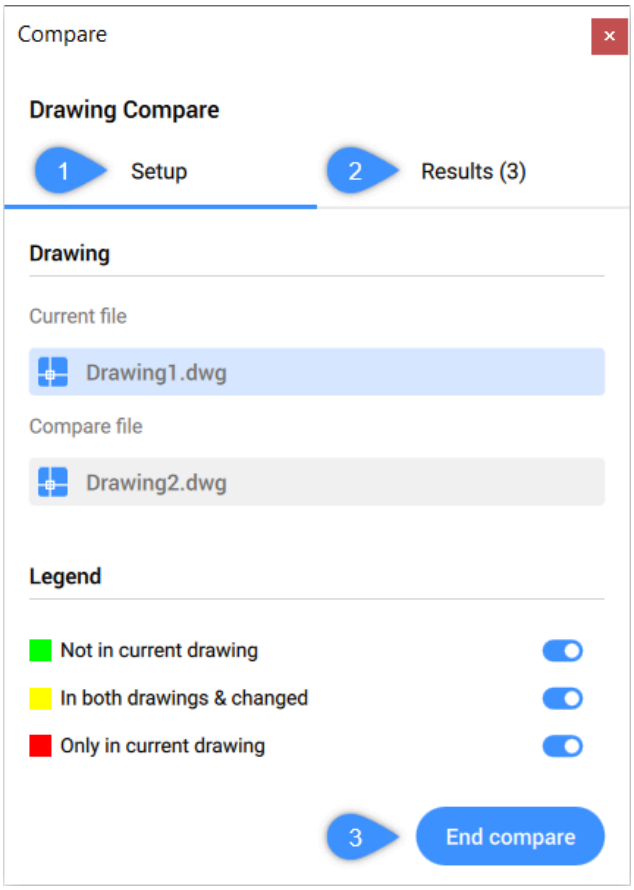
Opens the **Compare** panel.

Shape Lite Pro Mechanical BIM

9.137.1 Description

Opens the **Compare** panel to display it in the current workspace. The **Compare** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Compare** panel can be either floating, docked or stacked.

The **Compare** panel allows you to compare the current drawing with a selected drawing.

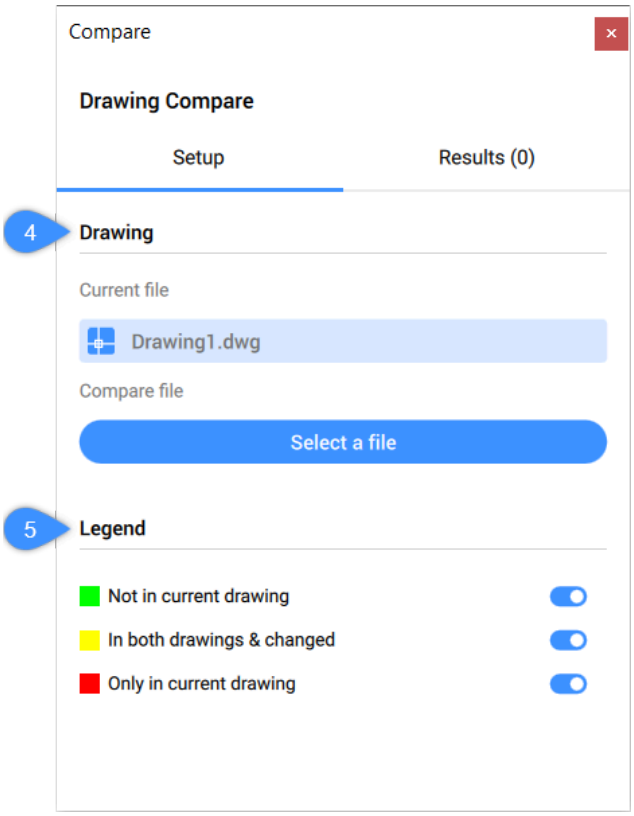


1 Setup

2 Results

3 End compare

9.137.2 Setup



4. Drawing

5. Legend

Drawing

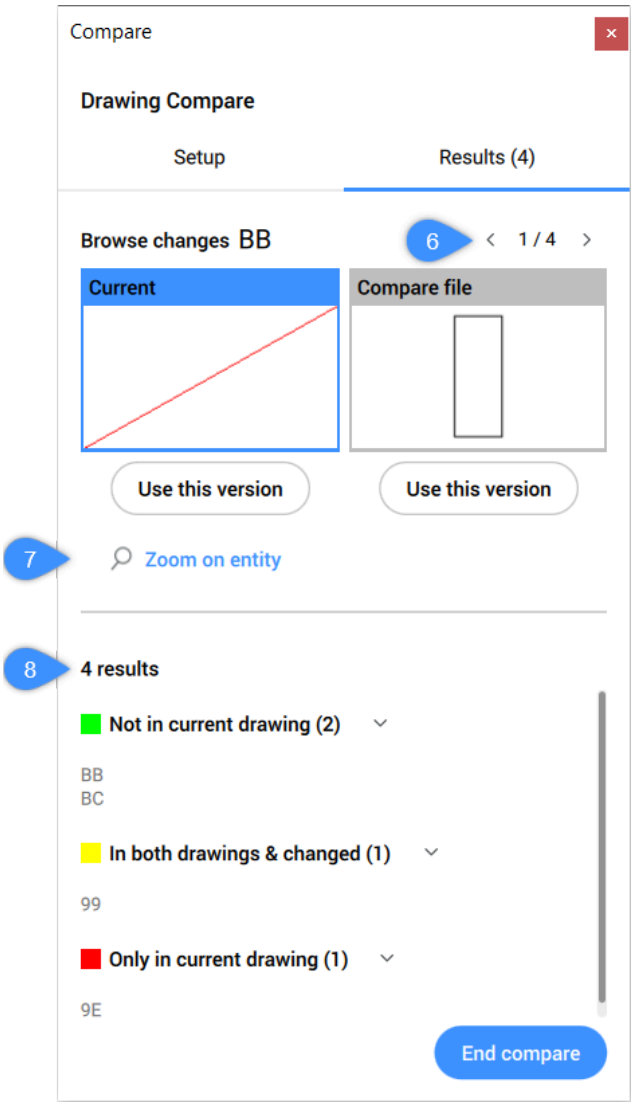
Displays the name of the current file and the selected file to compare the current file against. If no compare file is selected yet, click the **Select a file** button. The **Please pick a file to compare against.** dialog box opens to let you select a file.

Legend

Displays the colors in which the entities are displayed when the program finds differences. You can toggle the different categories on and off to determine whether the corresponding entities should be shown or not.

Note: You can click the colors to open the **Color** dialog box where you can pick another color for the selected category.

9.137.3 Results



6. Browse changes
7. Zoom on entity
8. Results list

Browse changes

Allows you to browse through the changes that are found between the two drawings. A preview of the entities is shown.

- Click on the arrows to browse through the changes
- Click on **Use this version** to select from which drawing you want to keep the entity

Zoom on entity

Zooms on to the entity that is shown in the preview

Results

Displays the handles of the entities that are different from the compare drawing.

- Not in current drawing: the entities only exist in the compare drawing
- In both drawings & changed: the entities exist in both drawings, but are changed

- Only in current drawing: the entities only exist in the current drawing

9.137.4 End compare

Ends the comparison and unloads the compare file.

9.138 -DWGHEALTH command

Combines the functionality of multiple stand-alone BricsCAD commands that help to improve overall drawing size and accuracy via command line.

Shape Lite Pro Mechanical BIM

9.138.1 Method

Runs one of the existing routine.

Note: Press any character to list available routines.

The predefined routines are:

2D cleanup
Performs the following tasks:

Purge
Checks and removes various types of unused items from the drawing. You can decide the object types that will be included in the check. You can run this check manually using the PURGE command.

Audit
Analyzes the integrity of the current drawing and fixes the errors. You can run this check manually using the AUDIT command.

Overkill - duplicate blocks
Checks the block definitions with identical contents, and (optionally) removes duplicates, according to the user's preferences. You can run this check manually using the OVERKILL command with the **Combine duplicate block definitions** option.

Blockify - match existing blocks
Checks for geometry that matches existing block definitions and replaces it with block references, decreasing file size and loading times, according to the user's preferences. You can run this check manually using the BLOCKIFY command with the **match existing Blocks** option.

3D cleanup
Performs the following tasks, already described:

- **Purge**
- **Overkill - duplicate blocks**
- **Blockify - match existing blocks**

And the below additional tasks:

DMAudit
Analyzes and automatically fix the problems in 3D geometry, accordingly with the user's preferences. You can run this check manually using the DMAUDIT command.

Blockify - equal solids
Replaces equal solids with block references to add structure to the drawing and decreases file size and loading times, according to the user's preferences. You can run this check manually using the BLOCKIFY command with the **Equal Solids** option.

Cleanup database
Performs the following tasks, already described:

- **Purge**
- **Overkill - duplicate blocks**

Fix errors
Performs the following tasks, already described:

- **Audit**
- **DMAudit**

9.139 DWGHEALTH command

Combines the functionality of multiple stand-alone BricsCAD commands that help to improve overall drawing size and accuracy.

Shape Lite Pro Mechanical BIM

Icon: 

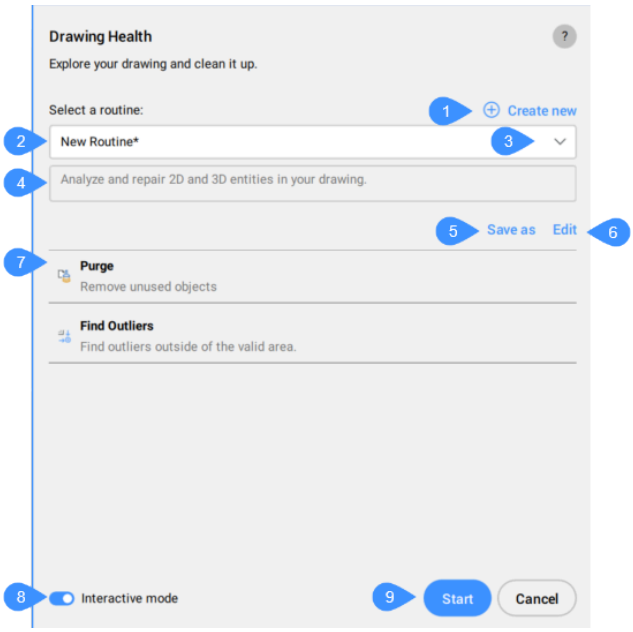
9.139.1 Description

Combines the power of multiple commands (PURGE, AUDIT, FINDOUTLIERS, SOLIDIFY, SIMPLIFY, OVERKILL, BLOCKIFY, OPTIMIZE, ARRAYDETECT) in one location. Reduces drawing size, removes unused styles, entities and layers and improves drawing quality.

You can run the DWGHEALTH command in the **Interactive mode**.

When you run the command for the first time, a tutorial dialog opens, explaining the DWGHEALTH command in two steps.

The DWGHEALTH command opens the **Drawing Health** Command Context panel:



1 Create new

2 Select a routine

3 Drop-down list

4 Description of the routine

5 Save as

6 Edit

7 List of the tasks included in the routine

8 Interactive mode

9 Start

9.139.2 Create new

Opens the **Create Routine** dialog box that allows you to add a new routine. See the related article **Create Routine dialog box**.

9.139.3 Select a routine

Allows you to select a routine from the drop-down list.

9.139.4 Drop-down list

Lists all available routines, predefined and custom.

The predefined routines are:

Fix errors
Analyze and repair 2D and 3D entities in your drawing.

Reduce drawing size
Reduce drawing size (where applicable) by removing unused objects, duplicate overlapping geometry, and block definitions with identical contents. Properties to be included or filtered can be selected under each task.

Repair 3D imports
Clean and repair imported 3D models from non-DWG programs. This routine addresses issues such as geometry errors, converting meshes to solids, and simplifying 3D solids after conversion. The routine may improve performance and reduce file size on large imported 3D solid files.

Click the **Manage routines...** button to open the **Manage Routines** dialog box. See the related article **Manage Routines dialog box**.

9.139.5 Description of the routine

Displays the description of the routine, defined in the **Edit Routine** dialog box.

9.139.6 Save as

Opens the **Save Routine** dialog box to save the custom routine with the desired name.

Note: The **Save as** button appears only after you run an unsaved routine via the **Run** button in the **Create Routine** dialog box or **Edit Routine** dialog box. See the related articles.

9.139.7 Edit

Opens the **Edit Routine** dialog box where you can edit the selected routine. See the related article **Edit Routine dialog box**.

9.139.8 List of the tasks included in the routine

Lists all tasks that will run with the selected routine.

9.139.9 Interactive mode

Switches to manual checks and fine-tuning. For each routine task, a panel will open containing the same options as the corresponding commands, where you can fine-tune the cleanup of the drawing.

9.139.10 Start

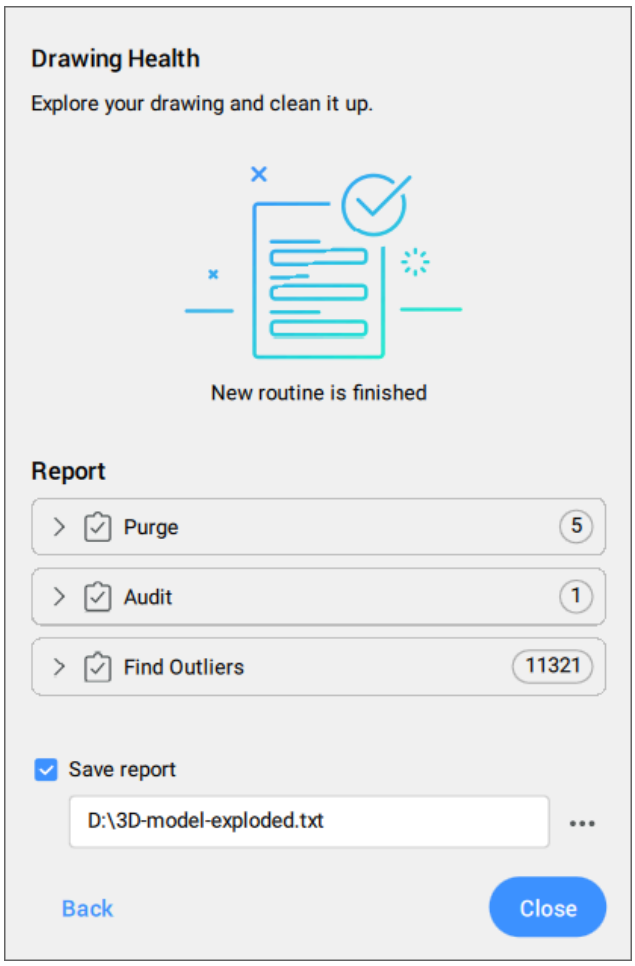
Performs the routine.

If the **Interactive mode** button is active, a panel opens for each routine task. Choose your settings and press the **Next** button to open the next task panel.

Note: DWGHEALTH tasks and their corresponding stand-alone commands support zoom functionality, where applicable. 'Applicable' here means that the task has something to zoom to (for example, purge doesn't). Zoom is available either by double-clicking an item or right-clicking it and selecting the **Zoom** option. In the latter case, the application will zoom to all selected objects, if applicable.

Note: For the **Overkill - duplicate blocks** task, when the Overkill operation touches multiple entities, it is only possible to apply the changes to the entire group.

After the last task, it opens the **Report** on the drawing's cleanup, which displays all items detected and cleaned.



Note: Tick the **Save report** check-box to save the report to a TXT file.

9.140 DWGPROPS command

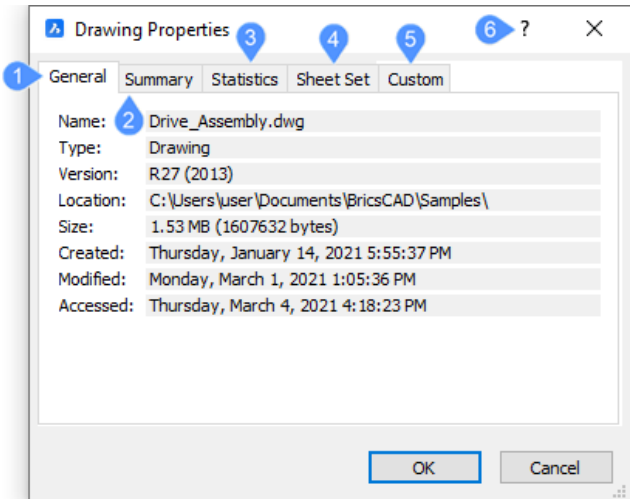
Opens the **Drawing Properties** dialog box.

Shape Lite Pro Mechanical BIM

9.140.1 Description

Opens the **Drawing Properties** dialog box.

The **Drawing Properties** dialog box allows you to view and modify general information about the drawing and user-defined properties.



1 General

2 Summary

3 Statistics

4 Sheet Set

5 Custom

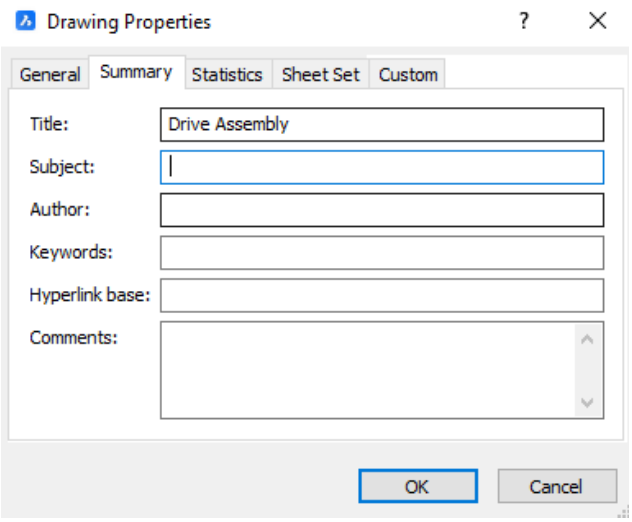
6 Command reference

9.140.2 General

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

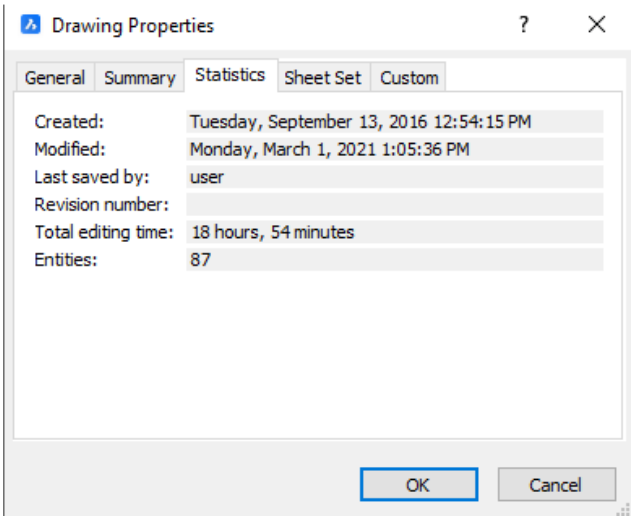
9.140.3 Summary

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.



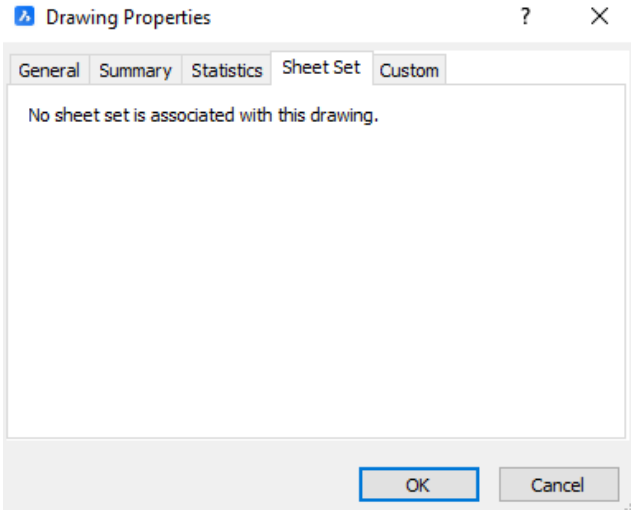
9.140.4 Statistics

Displays statistical information about the drawing, such as the creation and modification time.



9.140.5 Sheet Set

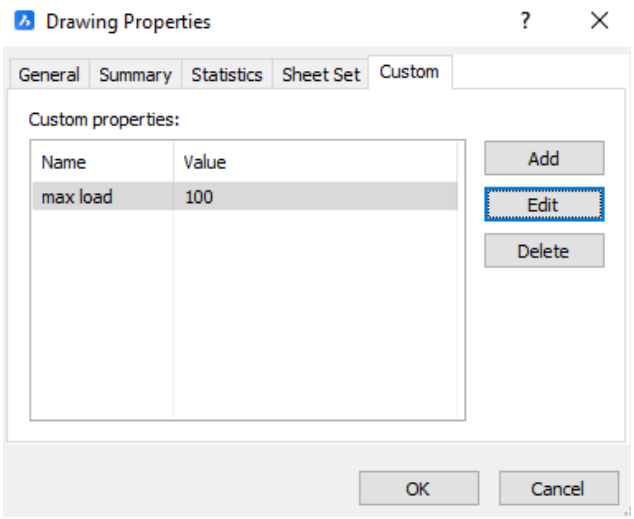
Displays sheet set data when the drawing is associated with a sheet set.



9.140.6 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.



9.140.7 Command reference

Opens the Bricsys Help article about the DWGPROPS command.

9.141 DXFIN command

Opens the **Load DXF file** dialog box.

Shape Lite Pro Mechanical BIM

9.141.1 Description

Opens the **Load DXF file** dialog box to select a DXF file to import into the current drawing.

9.142 DXFOUT command

Opens the **Select DXF file** dialog box.

Shape Lite Pro Mechanical BIM

Alias: DX

9.142.1 Description

Opens the **Select DXF file** dialog box to save data from the current drawing to a DXF file.

9.142.2 Options within the command

Enter decimal places of accuracy (0-16)
Specifies the accuracy of real numbers in the DXF file. Some CNC (computerized numerical control) machines require 4 decimal places in DXF files.

Entities
Specifies the entities to export. If unspecified, all entities are exported.

Binary
Specifies binary format for the DXF file. If unspecified, the DXF file is exported to ASCII (text) format.

Version

Specifies the DXF version number. If unspecified, the DXF file is exported to the most current version. Exporting to an older version can cause some entities or properties to be lost.

10. E

10.1 EATTEDIT command

Edits attributes contained in a single block.

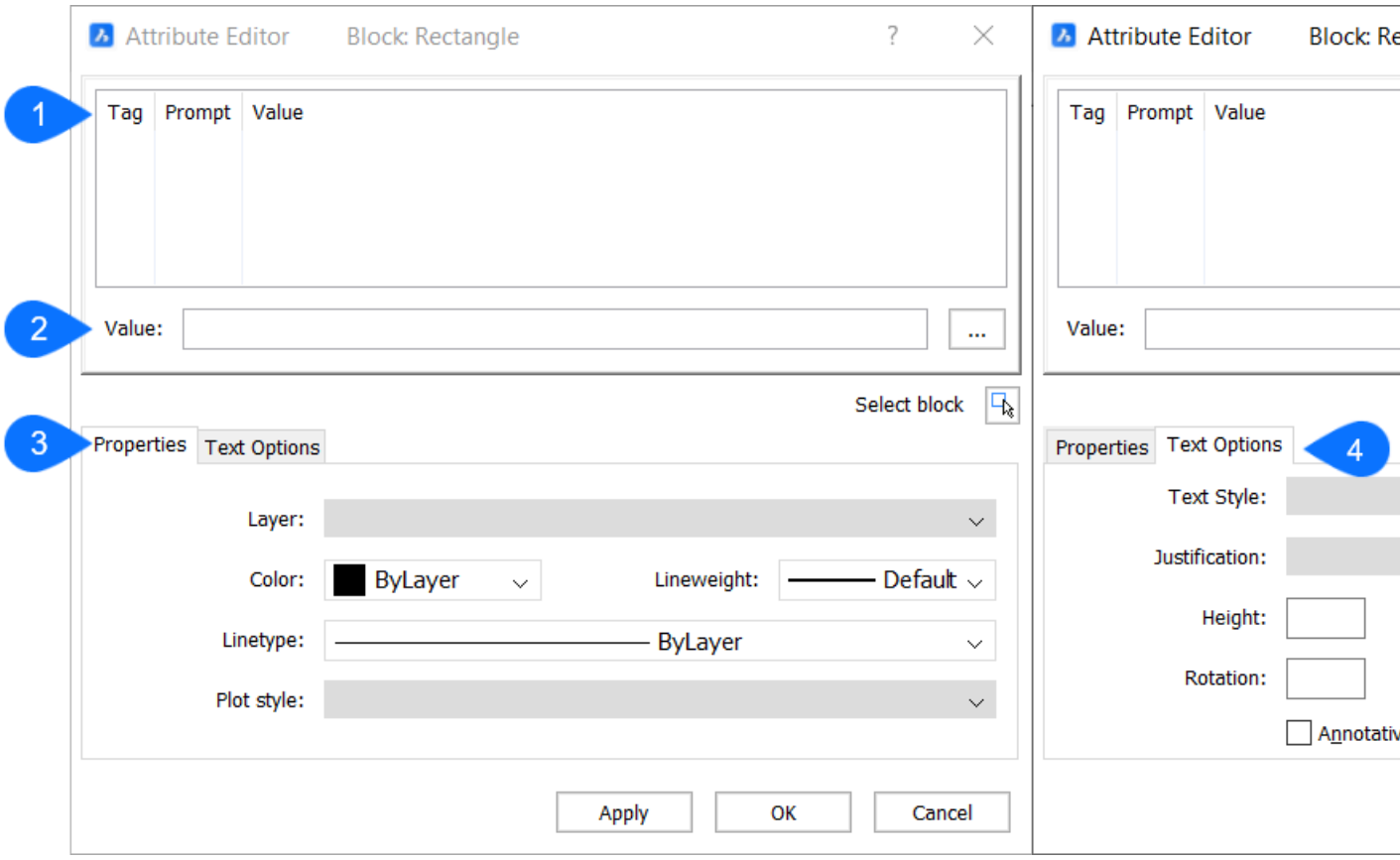
Shape Lite Pro Mechanical BIM

Icon:

Alias: ATE

10.1.1 Description

Edits the values and most properties of all attributes contained in a single block, through the **Attribute Editor** dialog box.



1 Overview list

2 Value

3 Properties

4 Text Options

10.1.2 Overview list

Displays an overview of all attributes used in the block.

10.1.3 Value

Displays the currently selected attribute from the list. A new value can be typed.

10.1.4 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.

10.1.5 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.

10.2 EDGESURF command

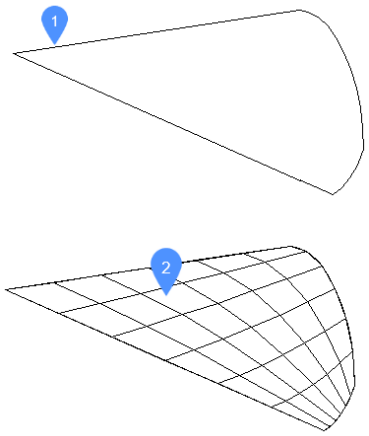
Creates 3D polygon mesh patches between four linear entities (short for “edge-defined surface”).

Shape Lite Pro Mechanical BIM

Icon: 

10.2.1 Method

Create an edge surface (2) by selecting each four connected linear entities (1) for it.



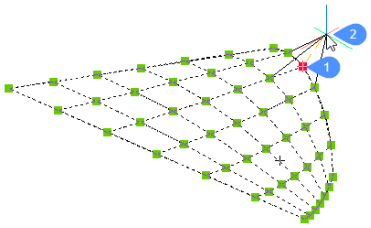
The four entities must be:

- Linear: open entities, such as lines, arcs, open splines, and open polylines.
- Connected: touching or overlapping, so that they form a closed area.

10.2.2 Grips Editing

Edge meshes can be edited directly through grips:

Drag a grip (1) to stretch (2) the adjacent faces.



10.3 EDITEDATA command

Edits extended entity data.

Shape Lite Pro Mechanical BIM

Icon:

10.3.1 Method

Select an entity to display the **Create or Edit Entity Data** dialog box.

10.4 EDITTIME command (Express Tools)

Tracks the active editing duration for a drawing.

Shape Lite Pro Mechanical BIM

10.4.1 Method

You can turn On and Off the timer, or reset it. The timing is suspended after a specified period of inactivity.

10.4.2 Options within the command

Reset
Allows you to reset the timer to zero. Timing is restarted automatically if the timer is running.

Timeout
Allows you to specify the period of inactivity, in minutes. The timer automatically suspends timing when the timeout period expires.

On
Allows you to start or resume the timer.

Off
Allows you to stop the timer.

10.5 ELEV command

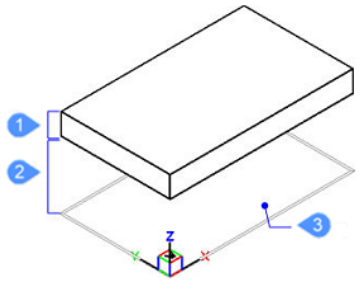
Changes the elevation and thickness of selected objects.

Shape Lite Pro Mechanical BIM

Icon:

10.5.1 Description

Changes the elevation (as measured from the x,y-plane in the z direction) and thickness (as measured from the elevation) of selected entities.



1 Thickness

2 Elevation

3 x,y-plane

10.5.2 Options within the command

New current elevation
Specifies the elevation above or below the x,y-plane. For a negative elevation, a negative value can be entered.

Note: The elevation is reported on the status bar as the z coordinate.

New current thickness
Specifies the thickness. Enter a negative value for thickness that are drawn downwards.

Note: The thickness begins at the elevation.

Note: The thickness is the extrusion distance, in the z direction, of non-3D entities, such as: points, lines, polylines, arcs and circles. When thickness is not 0:

- Points become vertical lines.
- Lines and open polylines become vertical planes.
- Arcs become curved surfaces.
- Circles become open-ended cylinders.
- Closed polylines become open-ended tubes.

10.6 ELLIPSE command

Creates an ellipse or elliptical arc.

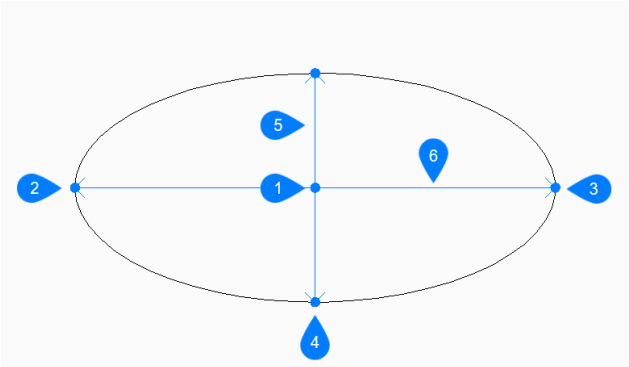
Shape Lite Pro Mechanical BIM

Icon:

Alias: EL

10.6.1 Description

Create an ellipse or elliptical arc from a combination of options including the center, axis endpoints, axis radius, rotation angle, and included angle.



1 Center
2 First end of ellipse axis
3 Second end of ellipse axis
4 End of other axis
5 Major axis
6 Minor axis

10.6.2 Method

This command has 3 methods to begin creating an ellipse:

- First end of ellipse axis
- Arc
- Center

If the PELLIPSE system variable is set to 1, the **Arc** option is not available.

10.6.3 Options within the command

First end of ellipse axis
Allows you to begin creating an ellipse by specifying a point on the ellipse's first axis.

Second end of axis
Specifies the end point of the ellipse's first axis.

Set Other axis
Specifies the radius of the other axis.

Rotation
Specifies the angle of the ellipse around the major axis. The angle can vary between 0 and 89.9. An angle of 0 degrees draws a circle. An angle of 89.9 degrees draws a very thin ellipse.

Arc
Allows you to begin creating an elliptical arc by specifying the first and the second ends of ellipse axes, and the radius of the axis.

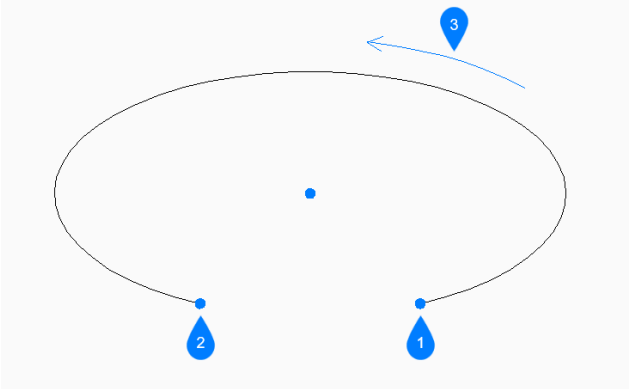
Set start angle of arc
Specifies the start angle to define the arc's starting point. The angle is measured counter-clockwise from the positive x-axis.

Parameter
Specifies values for the parametric formula that defines the elliptical arc:
p(b) = c + a * cos(u) = b * sin(u)
where:

- a = major axis
- b = minor axis
- c = center of the ellipse
- u = point along the arc

Included angle
Specifies the included angle, measured from the starting point.

End angle
Specifies the end angle to define the arc's endpoint.



1 Start angle

2 End angle

3 Included angle

Center
Allows you to begin creating an ellipse or elliptical arc by specifying its center point.

10.7 ENABLEASSOCVIEWS command

Adds the service data to the drawing and its Xrefs.

Shape Lite Pro Mechanical BIM

10.7.1 Description

It affects only associativity of the section result.

Note: If this command is not called and both GENERATEASSOCVIEWS and GENERATEASSOCATTRS are turned off, the result will be visually the same, but not associative.

10.7.2 Method

After the data is added, BIMSECTIONUPDATE and VIEWBASE commands cause no extra modifications in the referenced drawings.

Note: If the drawing contains a Xref file which requires modification, a message will appear to confirm modification of the Xref file.

10.7.3 Options within the command

Proceed
Toggles between Yes or No.

10.8 ENDCOMPARE command

Exits a drawing that was in comparison mode.

Shape Lite Pro Mechanical BIM

Icon:

10.8.1 Description

Exits a drawing that was in comparison mode begun by the DWGCOMPARE command.

10.9 ENDPOINT command

Toggles the **Endpoint** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



10.9.1 Description

Toggles the **Endpoint** entity snap to enable or disable snapping to endpoint. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

10.10 ERASE command

Erases entities from the drawing; erases the faces and edges of 3D solids.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DELETE, E

10.10.1 Method

Select entities, edges or faces to be removed.

Select the edges of an opening in a surface to delete the opening.

Select the edge(s) between coplanar faces of a 3D solid to remove the edges. Alternatively use DMSIMPLIFY command to remove redundant edges.

Select the interior faces of an opening in a 3D solid to remove the opening.

10.11 ETRANSMIT command

Opens the **Drawing Explorer** dialog box with **Dependencies** selected.

Shape Lite Pro Mechanical BIM

Icon: 

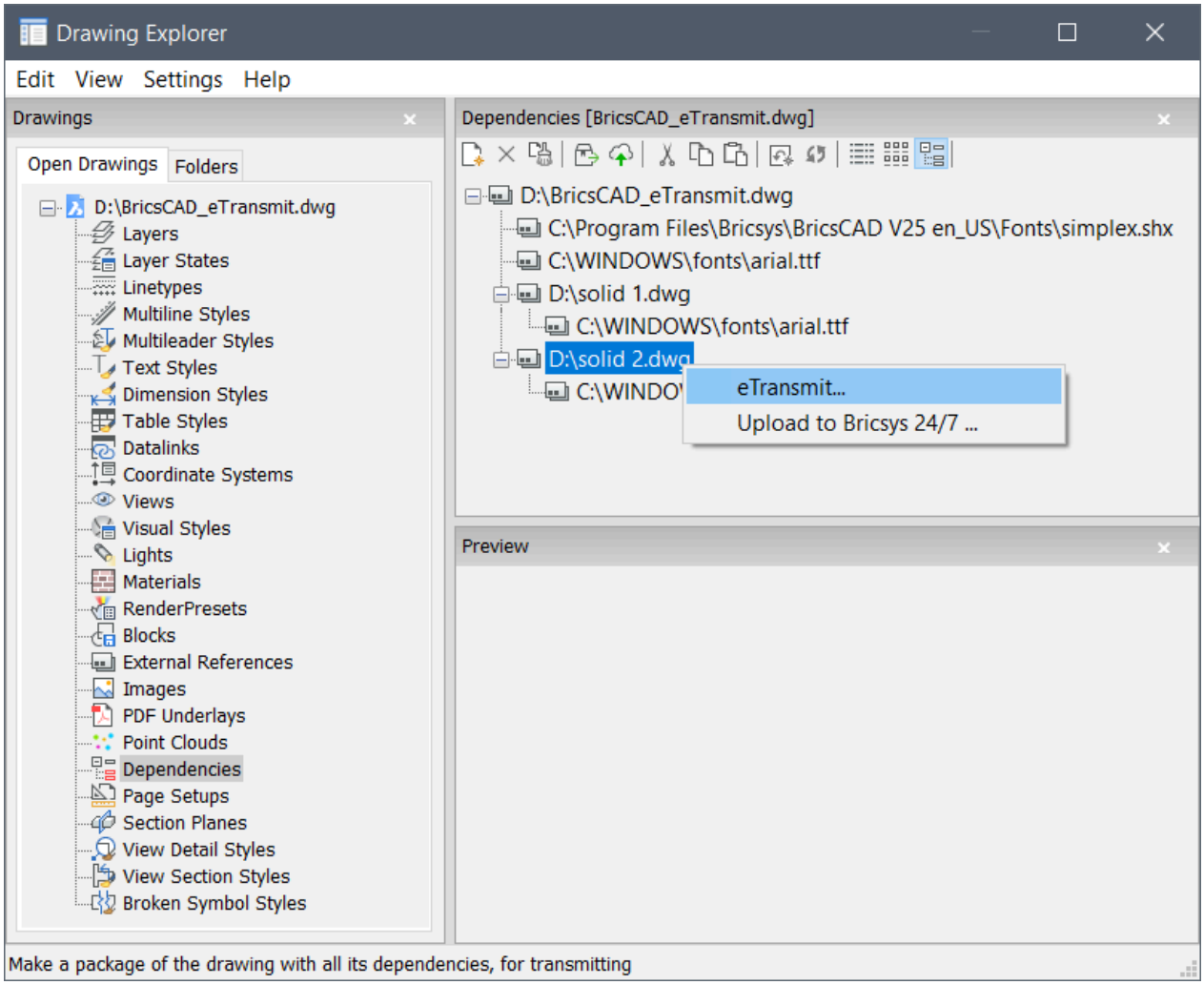
10.11.1 Description

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.

10.11.2 Methods

Opens the **Drawing Explorer** dialog box with **Dependencies** category selected to view and modify all dependencies in the current drawing.

Right-click an item listed in the **Dependencies** category to display the context menu options.



10.11.3 Context Menu Options

eTransmit

Starts the eTransmit procedure. The **eTransmit** dialog box opens, which lists the files to include (see **eTransmit dialog box** article).

Upload to Bricsys 24/7

Opens the **Upload to Bricsys 24/7** dialog box to allow you upload the drawing and its dependencies onto Bricsys 24/7 (see **Upload to Bricsys 24/7 dialog box** article).

10.11.4 Options within the command

New

Add another drawing to the eTransmit procedure. The **File** dialog box opens to let you choose another drawing.

Delete
Deletes Dependency definitions from the drawing.

10.12 -ETRANSMIT command

Creates a package ZIP file at the Command line.

Shape Lite Pro Mechanical BIM

10.12.1 Description

Creates a package ZIP file at the Command line, consisting the current drawing file and all its dependent files.

10.12.2 Options within the command

Create
Creates a package ZIP file that contains the drawing and its support files.

Yes
Automatically creates a package ZIP file that has the same name as the drawing. On subsequent usage of the command, the package will be saved in the previously used folder path.

No
Opens a **File** dialog box to let you select a folder and specify a file name.

Settings
Prompts you for changes to the settings in the Command line.

Save format
Specifies the format of the drawing files in the eTransmit package.

Output format
Specifies whether to create a ZIP file or copy the files to the output folder.

FOlder structure
Specifies how files are organized into folders.

FIle list
Determines which files are included in the transmittal package.

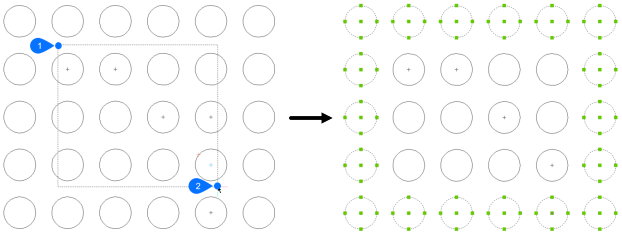
10.13 EXC command (Express Tools)

Selects all entities outside a specified rectangle.

Shape Lite Pro Mechanical BIM

10.13.1 Method

If you select a fist corner (1) and a second corner (2), a temporary rectangle is drawn. All entities in the drawing are selected, except the ones that are included in or intersect the selection window.



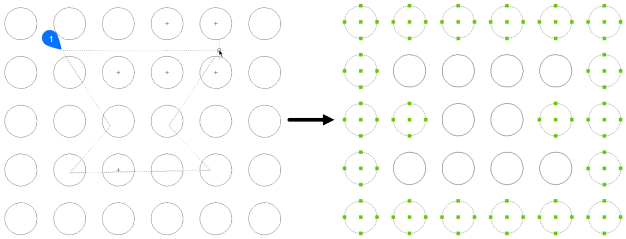
10.14 EXCP command (Express Tools)

Selects entities outside a specified polygon.

Shape Lite Pro Mechanical BIM

10.14.1 Method

Draw a temporary polygon (1), which is indicated with a dashed line. All entities in the drawing are selected, except the ones that are included in or intersect the selection polygon.



10.15 EXECUTETOOL command

Repeats the last tool used from the **Tool Palettes** panel.

Shape Lite Pro Mechanical BIM

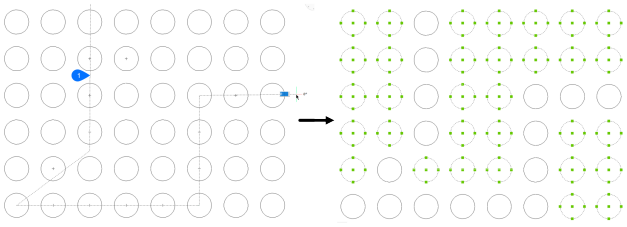
10.16 EXF command (Express Tools)

Selects all entities that do not intersect a specified fence line.

Shape Lite Pro Mechanical BIM

10.16.1 Method

Draw a fence line (1), which is indicated with a dashed line. All entities in the drawing are selected, except the ones that intersect the fence line.



10.17 EXOFFSET command (Express Tools)

Offsets the selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

10.17.1 Method

Enter the offset distance and select the entities to offset.

10.17.2 Options within the command

Through
Offsets the selected entity through a specified point.

Multiple
Offsets the selected entity multiple times.

Options
Lists the values of **Distance**, **Layer** and **Gaptype** in the command line.

Note: The **Gaptype** is controlled by the OFFSETGAPTYPE system variable.

10.18 EXP command (Express Tools)

Selects all entities in the drawing except the previous selection.

Shape Lite Pro Mechanical BIM

10.18.1 Description

The previous selection can be the entities from the last-used command or a set created by API calls.

10.19 EXPLAN command (Express Tools)

Displays the plan viewpoint of drawings, without changing the zoom factor.

Shape Lite Pro Mechanical BIM

Icon: 

10.19.1 Options within the command

UCS
Displays the plan view of a named UCS or Dynamic UCS after entering a name or placing the cursor over a 3D solid.

?
Lists the names of UCS's in the current drawing.

World
Displays the plan view of the World Coordinate System.

Current
Displays the plan view of the current UCS.

10.20 EXPBLOCKS command

Opens the **Blocks** category of the **Drawing Explorer** dialog box for inserting, deleting, and creating blocks.

Shape Lite Pro Mechanical BIM

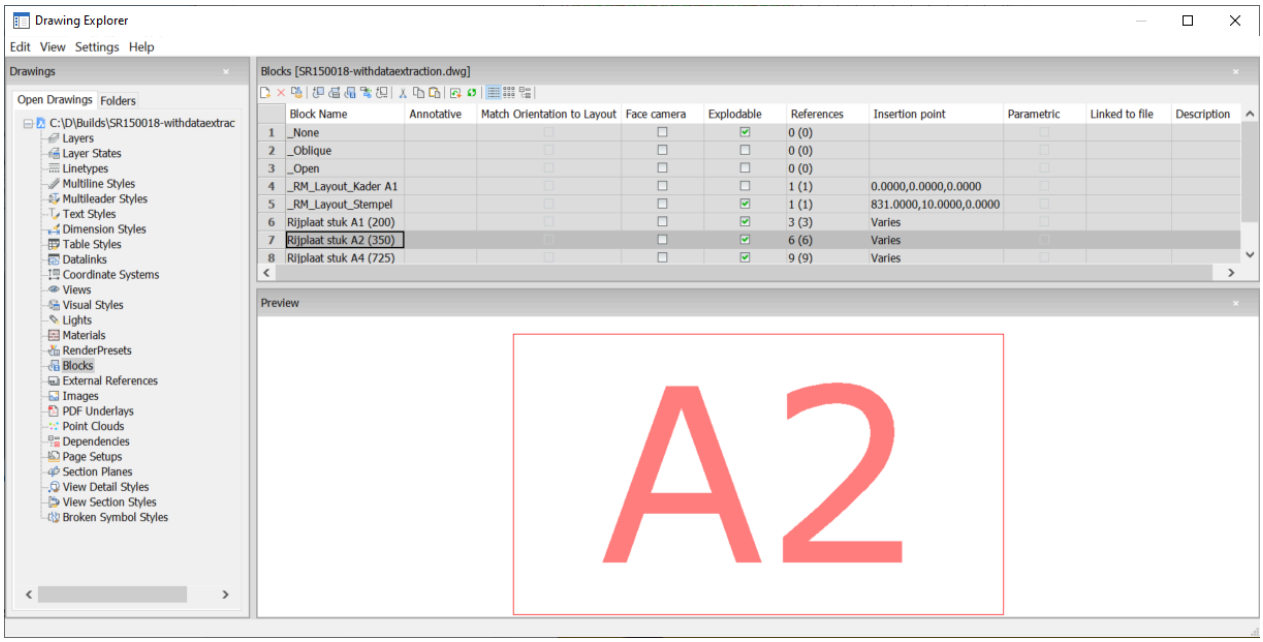
Icon:



Alias: XB

10.20.1 Description

Displays the **Blocks** category of the **Drawing Explorer** dialog box:



10.20.2 Context Menu Options

New

Creates new blocks; dismisses the **Drawing Explorer** dialog box, and the starts the -BLOCK command.

Delete

Deletes the block definition; available only for those block definitions not inserted in the drawing.

Purge

Removes unused block definition.

Insert

Inserts the selected block; dismisses the **Drawing Explorer** dialog box, and the starts the -INSERT command.

Insert External

Inserts DWG files as blocks; starts the INSERT command, and then displays the **Insert Block** dialog box.

Save Block

Exports the selected block as a DWG file; displays the **Save Block** dialog box.

Replace...
Opens the **Block Replace** dialog box to choose another block to replace.

Replace with Xref...
Opens the **Select an Xref file** dialog box to choose an Xref file to replace.

Add to Current Toolpalette
Adds the block to the current Tool Palettes.

Options...
Displays the **Drawing Explorer Options** dialog box to align, set the scale or the rotation of the inserted blocks.

Align with view
When checked, inserts the block to face the camera (blocks are rotated about their local Z-axis to face the camera).

Rename
Renames the selected block.

Note: Parametric blocks cannot be cut, copied or renamed.

Select All
Selects all blocks definitions.

Invert Selection
Inverts the selected blocks. For instance, if you have one block selected, then this option deselects it, and then selects all other blocks.

10.20.3 Options within the columns

Block Name
Specifies the name of the block.

Annotative
When checked, sets the **Annotative** property.

Match Orientation to Layout
When checked, sets the **Match Orientation** property of annotative blocks.

Face Camera
When checked, inserts the block to face the camera (blocks are rotated about their local Z-axis to face the camera). See the CAMERA command.

Explodable
When checked, blocks of this definition can be exploded into its component parts by the EXPLODE command. Non-Uniformly scaled blocks can be exploded only if EXPLMODE system variable is ON.

References
Reports the number of visible block inserts on all nested levels and, under the brackets, the total number of block inserts in the current drawing; the numbers are set by the program, so you cannot change it.

Note: The total number of block inserts includes block references on frozen or switched-off layers.

Insertion Point
Reports the insertion point of the block; reports "Varies" when the block has been inserted in the drawing more than once.

Parametric
Indicates if the block is parametric or not.

Linked to file
Shows file path for External Parametric blocks

Description
Optional description of the block.

10.21 EXPFOLDERS command

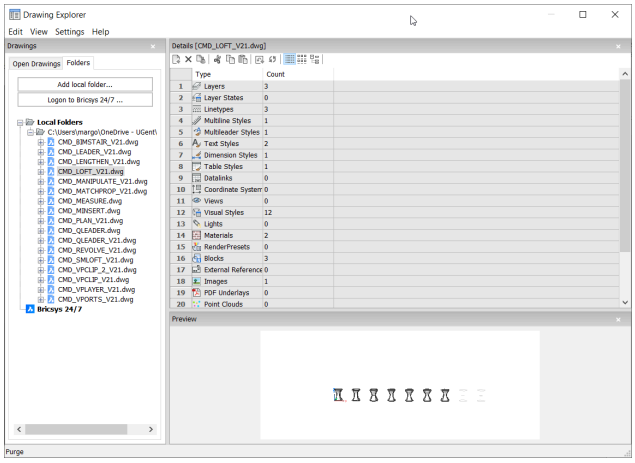
Opens the **Drawing Explorer** dialog box with the **Folders** tab selected.

Shape Lite Pro Mechanical BIM

Icon:

10.21.1 Description

Opens the **Drawing Explorer** dialog box on the **Folders** tab.



Note: The categories (such as Layers, Blocks, etc.) can be copy-pasted from **Folders** tab to **Open Drawings** tab.

10.21.2 Context Menu Options

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

10.21.3 Options within Folders tab

Add local folder
Opens a **Choose a Folder** dialog box.

Logon to Bricsys 24/7
Displays the **Bricsys 24/7** dialog box to log on.

10.22 EXPIMAGES command

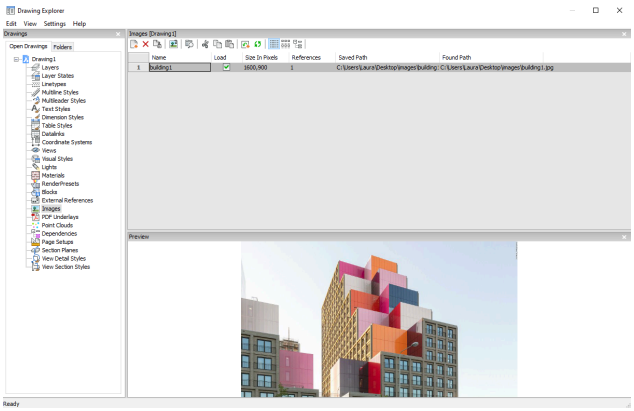
Opens the **Drawing Explorer** dialog box with **Images** selected.

Shape Lite Pro Mechanical BIM

Icon:

10.22.1 Description

Displays the **Images** category of the **Drawing Explorer** dialog box:



Supports the following formats:

BMP: Windows bitmap

ECW: Enhanced Compressed Wavelet format

GIF: CompuServe graphic image format

JPG, JPEG, and JPEG2000 (JP2, j2k): Joint Photographic Experts Group (common among digital cameras)

MTI: Multi-resolution Tiled Image (Bricsys raster format for large images)

PCX: PC Paintbrush

PNG: Portable network graphics

SID: MrSID is an acronym for Multi-resolution Seamless Image Database

TGA: Targa

TIF, TIFF: Tagged Image File Format

10.22.2 Context Menu Options

New
Attaches raster image files to the drawing. dismisses the **Drawing Explorer** dialog box, and the starts the IMAGEATTACH command. See IMAGEATTACH command.

Delete
Removes the image from the drawing. You may need to use the REGEN command to update the screen.

Insert
Inserts more images in the drawing, like the New button. Displays the **Attach Raster Image** dialog box. See IMAGEATTACH command.

Run extended search for missing attachments
Runs an extended search for missing attachments.

Clear cash
Clears the folder where temporary image cache files are stored.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the image located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.22.3 Options within the columns

Name
Specifies the name of the image. This usually is the image's file name.

Load
Toggles whether the image is displayed in the drawing:
On: displays the image in the drawing
Off: displays only the image's frame, which is turned off by the IMAGEFRAME command

Size in Pixels
Reports the size of the image in pixels, horizontally and vertically.

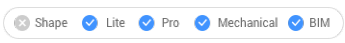
References
Reports the number of times the image occurs in the drawing.

Saved Path
Reports the path to the image file when first loaded. When the path cannot be found, click the **Browse** button which will display the **Choose a File** dialog box and locate the missing image file.

Found Path
Reports the current path to the image file. This path should match the Saved Path in most cases.

10.23 EXPLAYERS command

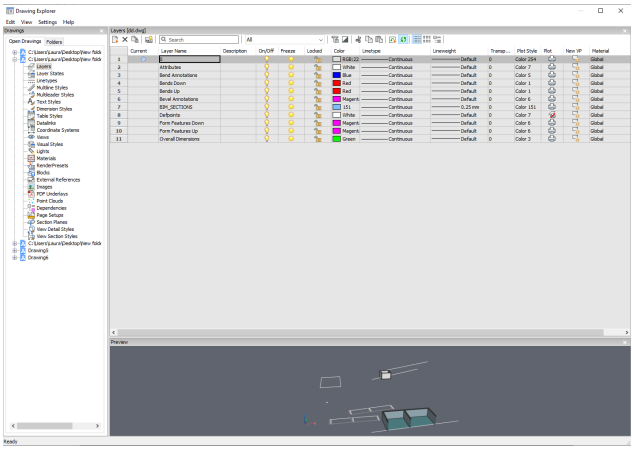
Opens the **Drawing Explorer** dialog box with **Layers** selected.



Icon:

10.23.1 Description

Opens the **Drawing Explorer** dialog box with **Layers** category selected to view and modify layers in the current drawing.



Note: When you open the **Drawing Explorer - Layers** dialog when in paper space, additional columns are displayed: **VP Freeze**, **VP Color**, **VP Linetype**, **VP Lineweight**, **VP Transparency**, and **VP Plot Style**, where VP is short for viewport.

Note: By default, these columns match the corresponding model space settings, but viewport layer properties can be set differently for each layout and for each paper space viewport. If the SHOWLAYERUSAGE variable is On, then the **Layer Usage** icons indicates whether viewport settings for the current layout and paper space viewport are different from model space settings:

- : current layer with viewport overrides.
- : layer with viewport overrides.
- : empty layer with viewport overrides.

Note: If no paper space viewport is active, then the settings apply to entities in the current layout. If a viewport is active, then viewport settings override the BYLAYER properties in the current viewport.

Note: The **VP Freeze** setting controls the Thaw/Freeze status of layers for the current layout tab or viewport. The **New VP** setting specifies the default Thaw/Freeze status of layers for new layout viewports.

10.23.2 Context Menu Options

New

Create a new layer definition into the drawing. The new layer that is created copies the properties of the currently selected layer.

Delete

Deletes layer definitions from the drawing. The following layer definitions cannot be deleted:

- Layer 0

- Defpoints
- Layers in use
- The current layer

Merge to…
Opens the **Merge Layers** dialog box. Here you can choose a target layer where the selected layers will be merged to.
The following layer definitions cannot be merged:

- Layer 0
- Defpoints
- The current layer

Note: you can only merge layers which are in use. If there aren't any entities assigned to a layer, the merge option will delete the selected layer.

Remove from group
Removes layers from the group they are assigned to.

Create Group Filter From Selection
Creates a group with the selected layers assigned to it.

Rename
Renames the selected layer.
Following layers can not be renamed:

- Layer 0
- Defpoints

Select All
Selects all layers.

Invert selection
Deselects the current selection and vice versa.

Remove Viewport Overrides
Removes overrides from properties that were set differently for viewports.

From Selected Layers

- For current viewport: removes overrides from the current viewport
- For all viewports: removes overrides from all viewports in the drawing

From All layers
Removes the overrides form all layers in the drawing for either the current viewport or all viewports.

Isolate Selected Layers
Dims the entities on all which are not selected. Use the LAYUNISO command to unisolate layers.

10.23.3 Options within Filters panel

Filters
You can think of layer filters as "layer groups", groups of layers that you need to show or turn on-or-off all at once.

New Properties Filter
Creates a new properties filter. Displays the **Layer Property Filter** dialog box (see **Layer Property Filter dialog box** related article).

New Group Filter
Creates a new group filter.

Layers state
Toggles the layers state of all the layers in the selected group.

- On: toggles the layers on.
- Off: toggles the layers off.
- Thaw: thaws the layers.
- Freeze: freezes the layers.
- Lock: locks the layers.
- Unlock: unlocks the layers.

Isolate layers
Isolates the layers in the selected group.

- Current viewport: removes overrides from the current viewport.
- All viewports: removes overrides from all viewports in the drawing.

Add layers from selection
In the selected layer group filter, the layers from the selected objects in the drawing are added.

Replace with layers from selection
The layers from the selected group filter are replaced with the layers of the selected objects in the drawing.

Add layers from panel selection
Adds the layers which are selected in the layers panel.

Paste with Layers
Pastes the copied or cut group, including its layers.

Convert to Group
Converts a property filter to a group filter.

10.24 EXPLODE command

Reduces complex entities to simpler ones.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: X

10.24.1 Description

Reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs.

Note: Entities on frozen and locked layers are not exploded.

10.24.2 Method

The result depends on the entities you selected:

- Polylines become lines and arcs. Width is lost.
- Regions become lines, arcs and splines.
- Dimensions becomes lines, text, and solids (arrowheads).
- Blocks become individual entities and nested blocks. You may need to use Explode a second time.
- ***Note:*** Only blocks that have Explodable property set can be exploded. Non-Uniformly scaled blocks can be exploded only if EXPLMODE=ON.
- ByBlock colors and linetypes may change.
- Attributes revert to attribute definitions.
- 3D surface entities become 3D faces.
- Meshes become 3D faces.

Note: Simple entities, such as line and circles, cannot be exploded.

10.25 EXPLORER command

Opens the **Drawing Explorer** dialog box.

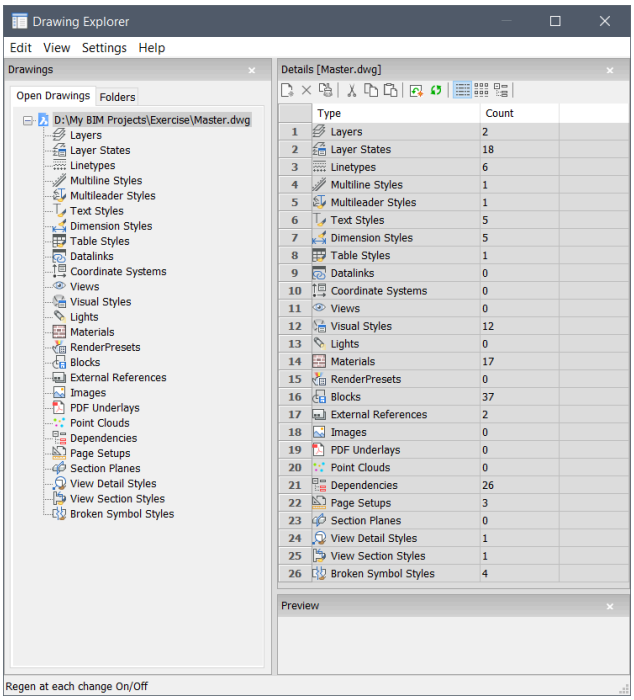
Shape Lite Pro Mechanical BIM

Icon: 

10.25.1 Description

Opens the **Drawing Explorer** dialog box on the last used tab or category to manage definitions and reference content that is used in the drawing.

Note: Press F1 to access help. To close, press X.



10.25.2 Options within the menu bar

View

Check boxes

Toggles the visibility of specific items in the **Drawing Explorer** dialog box on and off.

Tree/Icons Details

Selects a visual style to list all definitions of the selected drawing.

Regen

Regenerates all definitions.

Regen at each change

Regenerates definitions each time a change has been made.

Hide xref symbols

Toggles the visibility of xref symbols on and off.

Settings

Restore Default Layout

Restores the modified layout of the **Drawing Explorer** to the default layout.

Options...

Displays the **Drawing Explorer Options** dialog box. Options regarding inserting a block can be defined.

Columns

Columns can be switched on and off and repositioned by drag-and-drop:

- Move: drag it by its header to a new location.
- Change width: drag the separator line back and forth.
- Restore width: double-click the separator line.

Show all columns
Displays all columns including the ones that were hidden before.

Restore Column positions
Returns columns to their original position.

10.26 EXPORT command

Exports data from the current drawing to a variety of file formats.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DWFOUT

10.26.1 Description

Opens the **Export Drawing As** dialog box to save the data from the current drawing to a variety of file formats.

These are the supported file formats for export, depending on the license level:

BricsCAD Lite

- **FBX binary file** (*.fbx)
- **FBX ASCII file** (*.fbx)
- **Lithography** (*.stl)
- **Collada file**(*.dae)
- **Windows Meta File**(1)(2) (*.wmf)
- **Enhanced Windows Meta File**(1)(2) (.*emf)
- **AutoCAD Drawing** (*.dwg)
- **AutoCAD ASCII DXF** (*.dxf)
- **Drawing Template** (*.dwt)
- **Bitmap file**(2) (*.bmp)
- **3D DWF v6.01** (*.dwf)
- **Binary DWF v6.0** (*.dwf)
- **Zipped ASCII Encoded 2D Stream DWF v6.0** (*.dwf)
- **Compressed DWF v5.5** (*.dwf)
- **Binary DWF v5.5** (*.dwf)
- **ASCII DWF v5.5** (*.dwf)
- **Compressed DWF v4.2** (*.dwf)
- **Binary DWF v4.2** (*.dwf)
- **ASCII DWF v4.2** (*.dwf)

- **XPS DWFx** (*.dwfx)
- **Scalable Vector Graphics** (*.svg)
- **Micro Station DGN File** (*.dgn)

BricsCAD Pro

- **Unreal Datasmith** (1)(2)(*.udatasmith)
- **Rhino file**(1)(2) (*.3dm)
- **3D PDF** (*.pdf)
- **Assembly ACIS file** (*.asat)
- **Assembly ACIS file** (*.asab)

Note:

- Native export to 3D PDF and ASAT formats also support mechanical blocks, external references and solids.
- EXPORT stores a scale factor in the SAT and SAB files relative to mm which will be used to scale while IMPORT, and creates the current SPATIAL ACIS format which cannot be imported into AutoCAD®.

BricsCAD BIM / Mechanical / Ultimate

- **IFC2x3 file** (*.ifc)
- **IFC4 Reference View file** (*.ifc)
- **IFC4 Design Transfer View file** (*.ifc)
- **IFC4x1 file (deprecated)** (*.ifc)
- **IFC4x3 file** (*.ifc)
- **Revit Project** (*.rvt)

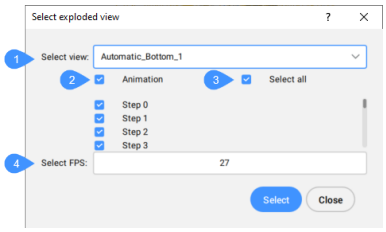
Note:

- BricsCAD BIM is officially certified for **IFC4 Reference View** (Architecture).
- The **IFC4x3 file** (IFC 4.3.2.0) export format is available only when the **Experimental mode** is enabled.
- The **IFC4x1 file (deprecated)** will be removed in a future BricsCAD release.

(1) Not available for BricsCAD for Linux.

(2) Not available for BricsCAD for macOS.

Note: If the EXPORT3DPDFWRITER system variable is set to 1 (Internal 3D PDF writer) and you save the drawing as a 3D PDF file, the **Select exploded view** dialog box displays:



1 **Select view**: Allows you to select from the drop-down menu an **Exploded view** to export. Select **No exploded view animation** to export the document without animation.

2 **Animation**: Toggles the export of exploded view with animation.

3 **Select all**: Selects all steps of the exploded view.

Note:

- Steps can be individually selected.
- Selected steps of exploded views are exported to 3D PDF as separate pages.

4 **Select FPS**: Sets the required number of frames per second (fps).

Note:

- When IFCEXPORTIDSPROPERTIESONLY system variable is ON, only the properties from the IDS-XML are exported.
- Civil objects (surfaces, alignments) can be exported to IFC4x3 files.
- Export to **Binary DWF v6.0** (DWF file) takes into account only the **Plot area** of the layout or model space defined in the Page Setup. Therefore, everything outside the plot area will be removed and or clipped away.
- The EXPORTMODELSPACE, EXPORTPAPERSPACE and EXPORTPAGESETUP system variables currently are dummies for DWF exports.
- More 3D formats are available through a separate module, Communicator for BricsCAD®. You can download it from the Bricsys website.

  The import and export procedures using Communicator for BricsCAD® are controlled through a series of user preferences. See the **Communicator** section in the **Settings** dialog box.
- To export to a PDF format, use the EXPORTPDF command.
- The EXPORT command supports components architecture based on regular blocks.
- Geometry-only is available for export to RVT format.
- When export the model as a Revit® project (RVT file), the entities are exported as Direct Shapes with classification.

10.27 EXPORTLAYOUT command

Opens the **Export layout to drawing** dialog box.

Shape Lite Pro Mechanical BIM

10.27.1 Description

Opens the **Export layout to drawing** dialog box to save data from the current drawing to a .dwg file. The current layout specifies what data to include in the drawing. After the drawing is created, a BricsCAD dialog box allows you to choose if you want to open the new drawing.

10.28 EXPORTPDF command

Saves data from the current drawing to a PDF file.

Shape Lite Pro Mechanical BIM

Icon: 

10.28.1 Description

Open the **Export drawing as** dialog box to save data from the current drawing to a PDF file.

Note: This command adds a watermark to export output obtained using an Academic license.

10.29 EXPORTSVG command

Generates SVG files for each step of the selected exploded view.

Shape Lite Pro Mechanical BIM

10.29.1 Method

Select an exploded view and press **Enter** key to open the **Select folder to save svg files** dialog box. Select a folder and all the steps of your exploded view are saved there in SVG format.

Note: When the selected output folder already contains SVG files with identical names, a warning message asking if you overwrite them appears in the Command line. If you choose No, select the output location again.

To select an exploded view, you can type either its name or its index in the Command line.

Note:

- Use the BMEXPLODE command to create exploded views.
- The files are generated according to the options available via SVGOPTIONS command.

If the **Default camera** property of an exploded view is enabled, and the **Camera** property of its initial step is **Default**, the original camera of the model will be used for the initial step (that is, the camera of the model space before generation started), and it will be used until the first step with the property **Camera** = **Custom**.

To fix this:

- Open the exploded view for editing with the BEDIT command.
- In the **Mechanical Browser** right-click the initial step and choose **Apply**.
- Choose the proper camera for the initial step.
- To do this you can rotate the view using RTROT command.
- Right-click the initial step and choose **Update camera view**.

Its **Camera** will become **Custom**.

- Save the block.

Note: This is not a bug, it is the expected behavior.

10.29.2 Options within the command

list exploded views (?)
Lists all exploded views from the current drawing, along with their indexes.

10.30 EXPORTTOAUTOCAD command

Converts AEC entities into native entities.

Shape Lite Pro Mechanical BIM

10.30.1 Description

Creates a new version of a drawing file with all AEC entities converted to blocks containing native entities.

Note: The EXPORTTOBRICSCAD and EXPORTTOAUTOCAD commands are identical.

10.30.2 Method

Define the file format, whether to bind Xrefs, the bind type, the file name prefix, and the file name suffix, or keep the default settings.

If you enter a file name (not accepting the default one) in the command line (FILEDIA=0), the output file will be created in the same folder as the original drawing, not the current working folder.

10.30.3 Options within the command

Format
Sets the file format of the new file. The default value is **2018**.

Bind
Decide whether to bind the Xref files to the new drawing or not. The default value is **Yes**.

Yes
Binds the Xref files to the new drawing. Layers and other Xref-dependent named entities are merged into the new drawing.

No
Keeps the Xrefs as links to other drawings.

bind Type
Defines the behavior of Xref-dependent entities when the **Bind** option is set to **Yes**. The default value is **Insert**.

Bind
Keeps the names of layers and other Xref-dependent entities.

Insert
Incorporates the names of Xref-dependent entities into the new drawing while excluding the original file name.

Prefix
Specifies the new file name prefix.

Suffix
Specifies the new file name suffix.

10.31 EXPORTTOBRICSCAD command

Converts AEC entities into native entities.

Shape Lite Pro Mechanical BIM

Icon: 

10.31.1 Description

Creates a new version of a drawing file with all AEC entities converted to blocks containing native entities.

Note: The EXPORTTOBRICSCAD and EXPORTTOAUTOCAD commands are identical.

10.31.2 Method

Define the file format, whether to bind Xrefs, the bind type, the file name prefix, and the file name suffix, or keep the default settings.

If you enter a file name (not accepting the default one) in the command line (FILEDIA=0), the output file will be created in the same folder as the original drawing, not the current working folder.

10.31.3 Options within the command

Format
Sets the file format of the new file. The default value is **2018**.

Bind
Decide whether to bind the Xref files to the new drawing or not. The default value is **Yes**.

Yes
Binds the Xref files to the new drawing. Layers and other Xref-dependent named entities are merged into the new drawing.

No
Keeps the Xrefs as links to other drawings.

bind Type
Defines the behavior of Xref-dependent entities when the **Bind** option is set to **Yes**. The default value is **Insert**.

Bind
Keeps the names of layers and other Xref-dependent entities.

Insert
Incorporates the names of Xref-dependent entities into the new drawing while excluding the original file name.

Prefix
Specifies the new file name prefix.

Suffix
Specifies the new file name suffix.

10.32 EXPORTLINECURVELABELS command

Exports line and curve labels to a CSV file.

Shape Lite Pro Mechanical BIM

Icon:

10.32.1 Description

Opens the **Select filename** dialog box to save the line and curve labels to a CSV file. Segment length, direction, grade, area, radius, external tangent, chord length, chord direction, sum length and tag number values are saved for all line and curve labels in the drawing.

10.33 EXPPDFS command

Attaches PDF files as underlays to the current drawing through the Drawing Explorer.

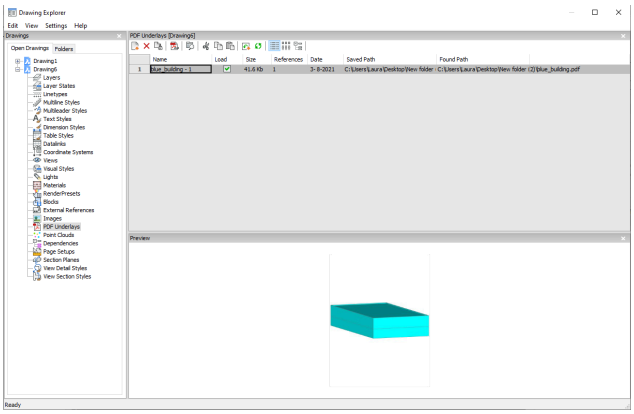
Shape Lite Pro Mechanical BIM

Icon:

Note: To import PDF files as drawing entities, use the PDFIMPORT command.

10.33.1 Description

Displays the PDF Underlays category of the **Drawing Explorer** dialog box:



10.33.2 Option within the command

Name
Reports the name of the PDF insert. This name is generated by the program, but can be edited by pressing double click on the name or right-click and choose Rename from the shortcut menu.

Load
Toggles loading of the PDF:
On: loads the PDF, and makes it visible.

Off: unloads the PDF, making it invisible.

Size
Reports the size of the PDF file.

References
Reports the number of times the PDF is attached to the drawing.

Date
Reports the date of the file. This can be useful in determining whether you ware working with the most recent revision.

Saved Path
Reports the path to the PDF file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing PDF file.

Found Path
Reports the current path to the PDF file. This path should match the Saved Path in most cases.

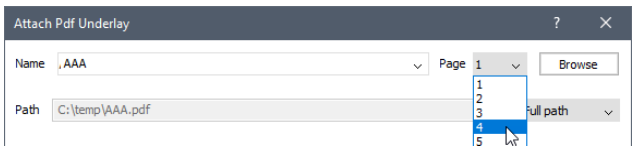
10.33.3 Context menu options

New
Attaches a PDF file to the current drawing. From the Select PDF Underlay File dialog box open a PDF file, specifies the parameters in the Attach PDF Underlay dialog box and an insertion point to locate the lower-left corner of the PDF.

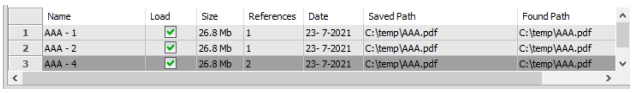
Delete
Detaches the selected PDF attachments without warning.

Purge
Purges selected PDF files from the drawing; available only for those PDF file definitions that are not inserted in the drawing.

Insert
Attaches additional pages from a multi-page PDF file already attached to the current drawing.
From the **Attach Pdf Underlay** dialog box choose a page number from the Page drop-list.



The added page appears in Drawing explorer with the page number as a suffix to its name.



Run extended search for missing attachments
Runs an extended search for missing attachments.

Clear cache
Clear Cache for PDFs.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the PDF file located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.34 EXPRESSMENU command (Express Tools)

Activates the EXPRESSTOOLS menu and ribbon.

Shape Lite Pro Mechanical BIM

10.35 EXPRESSTOOLS command (Express Tools)

Activates the Express Tools.

Shape Lite Pro Mechanical BIM

10.36 EXPUCS command

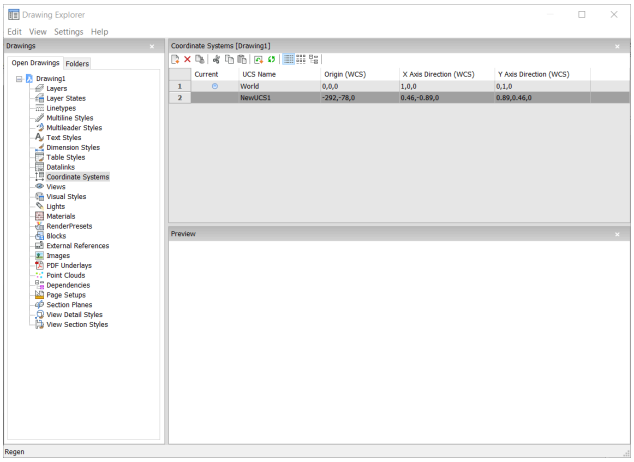
Creates, renames and deletes named UCS's through the **Drawing Explorer** dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDUCS, UC

10.36.1 Description

Opens the **Drawing Explorer** dialog box on the **Coordinate systems** category:



10.36.2 Options within the columns

Current
Indicates the current UCS.

UCS Name
Lists the names of UCS's in the drawing; click to rename.

Origin (WCS)
Reports the origin (0,0,0) of the UCS in x,y,z coordinates of the WCS; cannot be edited.

X Axis Direction (WCS)
Reports the direction of the x axis in WCS coordinates.

Note: Cannot be edited.

Y Axis Direction (WCS)
Reports the direction of the y axis in WCS coordinates.

Note: Cannot be edited.

10.36.3 Context Menu Options

New
Creates additional UCS into the drawing through the model space.

Delete
Deletes Coordinate system from the drawing. The World coordinate system definitions cannot be deleted.

Rename
Renames the Coordinate System.

Select All
Selects all Coordinate System definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected Coordinate System as current.

10.37 EXPXREFS command

Combines several xref-related commands through the **Drawing Explorer**.

Shape Lite Pro Mechanical BIM

Icon: 

10.37.1 Description

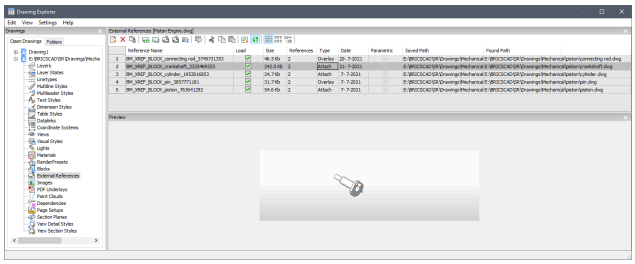
This command attaches DWG files to the current drawing and controls the state of the attachments (short for "external reference").

Note: The XDWGFADECTL system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Note: The BINDTYPE system variable influences the behavior of the Bind option of the -XREF command.

10.37.2 Method

Displays the External Reference section of the **Drawing Explorer**:



10.37.3 Option within the command

Reference Name

Reports the name of the xref insert. This name is assigned by the program, but could be edited by pressing double click on the name or right-click and choose the Rename from the shortcut menu.

Load

Toggles loading of the xref.

On: loads the xref, and makes it visible

Off: unloads the xref, making it invisible

Size

Reports the size of the xref DWG file. Very large files can slow down the system.

References

Reports the number of times the xref is attached to the drawing.

Type

Reports whether the xref is attached or overlaid, how they treat xrefs that have xrefs themselves:

Attach: all xrefs are displayed.

Overlay: only the first xref of nested xrefs is displayed.

Date

Reports the date of the file. This can be useful in determining whether you ware working with the most recent revision.

Parametric

Indicates if the Xref drawings is parametric or not.

Saved Path

Reports the original path to the xref DWG file when first loaded. When the path cannot be found, click the Browse button which will display the **Choose a File** dialog box and locate the missing DWG file.

Path for parametric blocks cannot be changed.

Found Path

Reports the current path to the xrefs DWG file. This path should match the Saved Path in most cases.

10.37.4 Context menu options

Attach Xref

Attaches DWG files as xrefs to the current drawing. From the **Attach External Reference** dialog box browse and choose a dwg file and specify the parameters and an insertion point to locate the xref file. See -XREF command.

Detach Xref
Erases the selected xref from the drawing without warning. This is equivalent to using the Detach option.

Reload Xref
Reloads the selected xref. This is useful when the original DWG file has changed and you want to see the updated version in your drawing.

Unload Xref
Unloads the selected xref. This hides the xref from view. You can also click the checkboxes in the Load column.

Bind Xref
Binds the xref to the current drawing to make the xref part of the drawing. References to the xref disappear from the **Drawing Explorer**.

Note: This option is not available for xrefs that are unloaded.

This option sets the BINDTYPE system variable OFF (Traditional binding behavior), thus influencing the behavior of the Bind option of the -XREF command.

Insert Xref
Converts xrefs into blocks. This is like using the Insert command to insert external DWG files in drawings. References to the xref disappear from the **Drawing Explorer**.

Note: This option is not available for xrefs that are unloaded.

Open Xref
Opens the xref DWG file for editing. See the XOPEN command.

Run extended search for missing attachments
Runs an extended search for missing attachments.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the Xref file located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.38 EXTEND command

Extends or trims one or more open entities to a boundary entity.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: EX

10.38.1 Method

Choose one or more entities to use as boundaries entity (2). These are the entities to which subsequently picked entities are extended or press ENTER to choose all entities in the drawing as boundaries. There is no need to pick individual entities as boundaries, except in certain cases.

Chooses the entity to be extended (1). When another end of an entity could be extended to a boundary, BricsCAD extends the closest end to your pick point.



1 Entity to be extended

2 Boundary entity

3 Extended entity

Note: Switches to trim mode: press and hold the Shift key to select the part of an entity to be trimmed at the intersection with the nearest boundary entities. See the TRIM command.

10.38.2 Options within the command

Bounding edges
Defines the bounding edges.

Fence
Allows fence selection, an irregular selection window.

Crossing
Allows crossing rectangle selection. The entities that cross the specified rectangle are extended.

Edge
Toggles edge mode.

Note: The EDGEMODE system variable also controls how boundary edges are checked.

Projection
Specifies how the entity is projected to the boundary.

Note: The PROJMODE system variable also controls the projection mode.

No projection
Extends only those entities that intersect real boundaries in 3D space.

xy plane of Ucs
Projects entities and boundaries to the x,y - plane of the current UCS, and then extends the projected entities that would intersect projected boundaries.

current View
Projects entities into the current view, and then extends them accordingly.

eRase
Deletes selected entities.

mOde
Defines the mode in which the command is acting.

Note: The TRIMEXTENDMODE system variable also controls the extend mode.

Standard
Acts in standard mode.

Quick
Acts in quick mode, in which all entities automatically act as bounding edges.

10.39 EXTENSION command

Toggles the **Extension** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



10.39.1 Description

Toggles the **Extension** entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

10.40 EXTRACTBLOCKS command

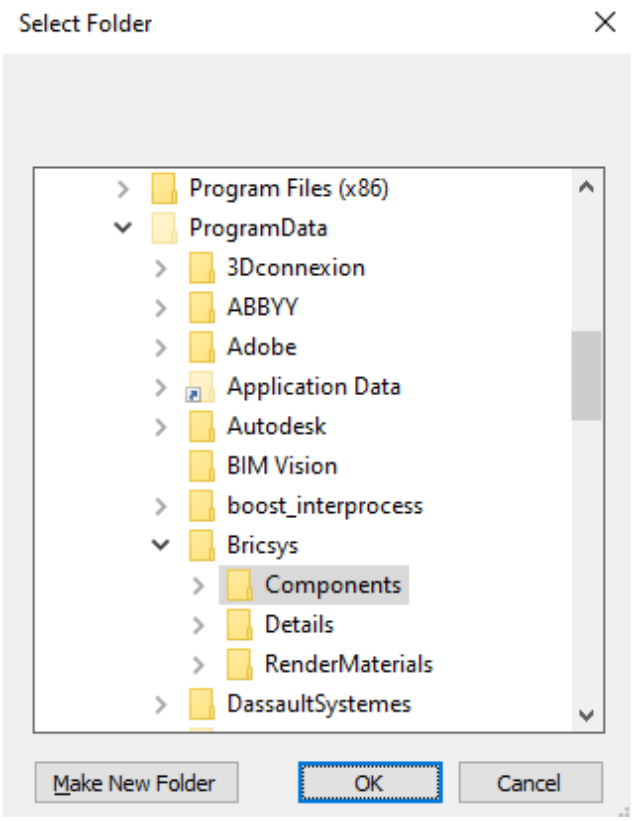
Extracts blocks to individual files.

Shape Lite Pro Mechanical BIM

10.40.1 Method

This command extracts a block instance from a drawing, creating a separate DWG file in a user-specified location.

After selecting a block to extract, the **Select Folder** dialog box displays to choose the location for the new created DWG file.



10.41 EXTRIM command (Express Tools)

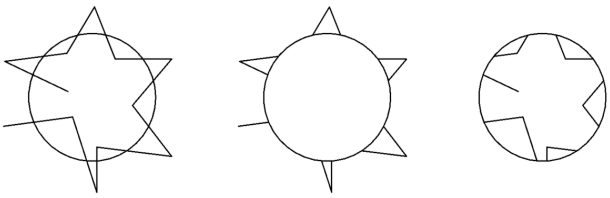
Trims entities that cross a cutting edge.

Shape Lite Pro Mechanical BIM

10.41.1 Method

Select the cutting edge entity, then click on the side you want to erase the entities from.

Note: The cutting edge entities accepted are: 3D Face, Polyline, Line, Circle, Arc, Ellipse, Image, Text or Attribute Definition.



10.42 EXTRUDE command

Creates 3D solids or surfaces by extruding entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: EXT

10.42.1 Description

Creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries.

Note:

- The SELECTIONPREVIEW system variable must be set to 2 or 3 to highlight faces.
- Depending on the value of the DELOBJ system variable, the source entities are either retained or deleted. If not, you are prompted whether you want the entities deleted or not.

10.42.2 Method

There are two methods to extrude entities:

- Create 3D solids.
- Create surfaces.

Note: You can extrude multiple profiles simultaneously.

10.42.3 Options within the command

MOde
Allows you to create either solids or surfaces.

SOlid
Creates 3D solids.

Note: Only for **Solid** mode, when the CREATESKETCHFEATURE system variable is ON, sketch features for extrude are created in a dedicated BC_SKETCHES layer, which is not visible by default. The sketch is visible and editable as block references in the **Mechanical Browser** panel. Also, the sketch features for extrude are visible and editable as block references in the **Mechanical Browser** panel.

Tip: Creating an extruded feature with shared faces from another solid is also possible.

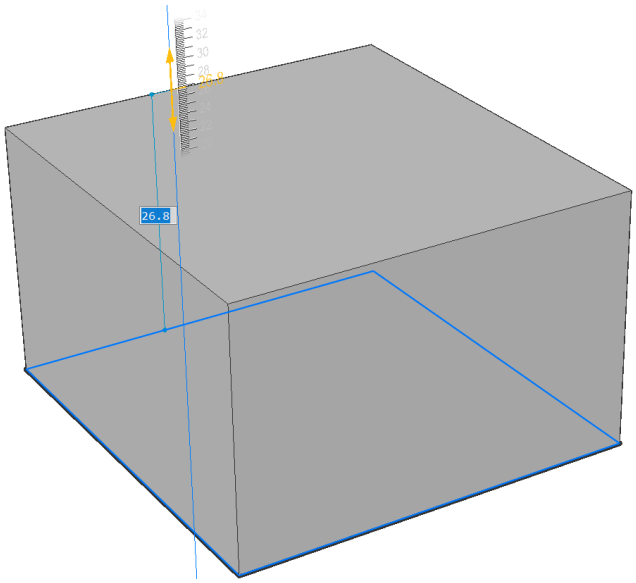
Note: The CREATESKETCHFEATURE system variable can also be controlled by pressing the **CreateSketchFeature** toggle button  in the ribbon.

SUrface
Creates surfaces.

Note: EXTRUDE does not extrude open 2D entities as 3D solids because they cannot be watertight.

Specify height of extrusion
Allows you to set a height of extrusion. The extrusion height is measured perpendicularly to the source entity.

Note: You can specify the height dynamically, using the **Manipulator**, or typing a distance value.



Direction

Allows you to specify the extrusion direction.

Path

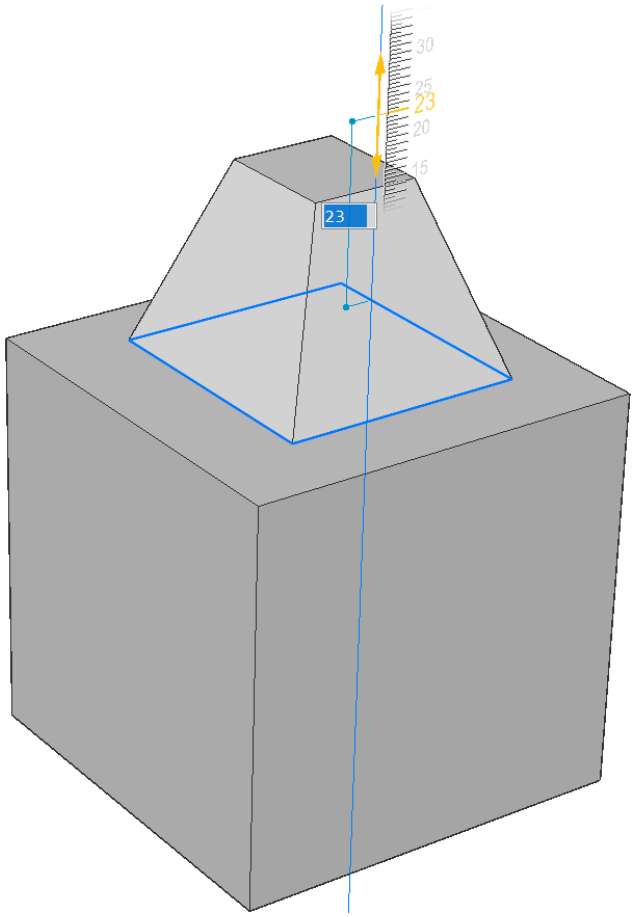
Specifies the extrusion height and taper by another entity. The program uses the path to determine how to extrude the source entity.

Note: The path entity cannot lie in the same plane as the extrusion entity.

Taper angle

Specifies the taper angle for the extrusion. The angle is measured from the extrusion direction. A negative value tapers outwards.

Note: The angle must be shallow enough so that the sloping sides do not intersect after the top of the extrusion.



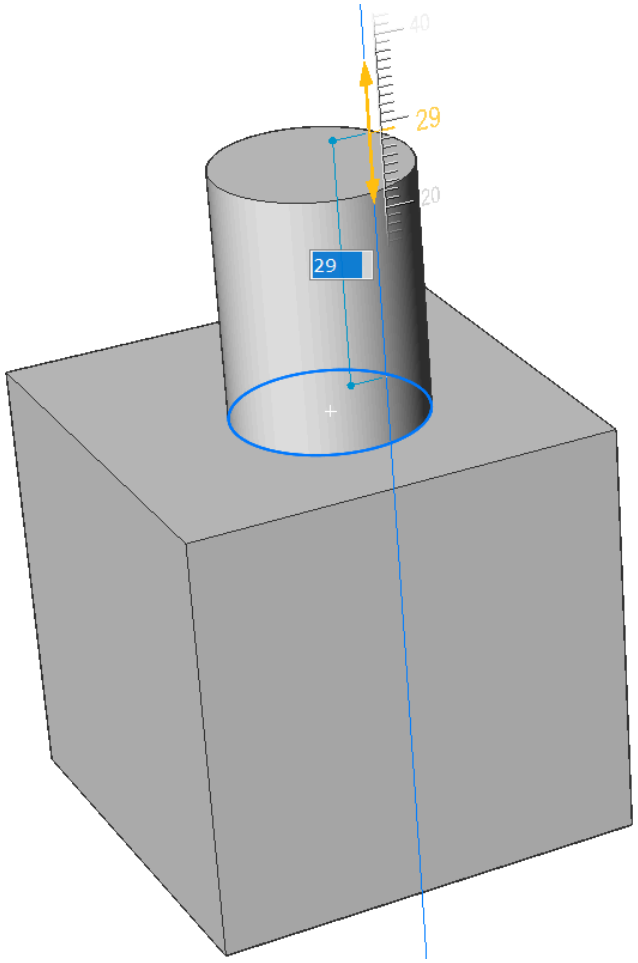
Auto

The result depends on the extrusion direction and the value of the four **Extrude mode** system variables: EXTRUDEOUTSIDE, EXTRUDEINSIDE, INTERSECTEDENTITIES and UNITESURFACES.

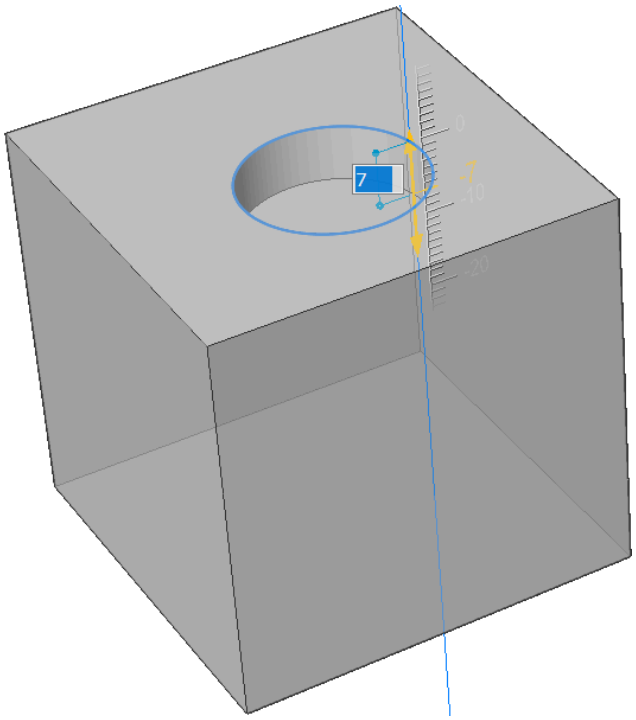
Note: The default value of all above **Extrude mode** system variables depends on the Workspace:

- **0** for Draft and Modeling.
- **1** for Mechanical and BIM.

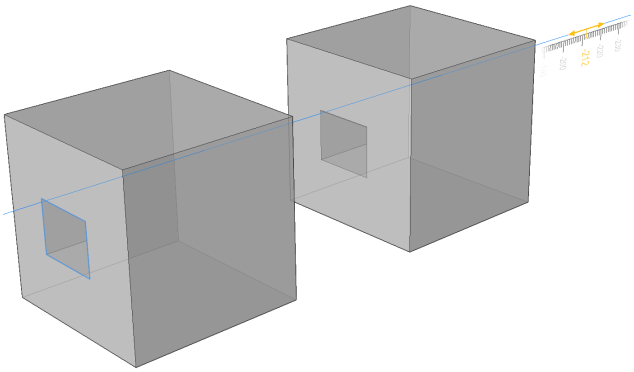
For example, in the Mechanical and BIM workspaces if all Extrude Mode system variables have the default value, the extrude result is like below:

- When you highlight an entity and extrude it to the outward, a new volume is added.
- 

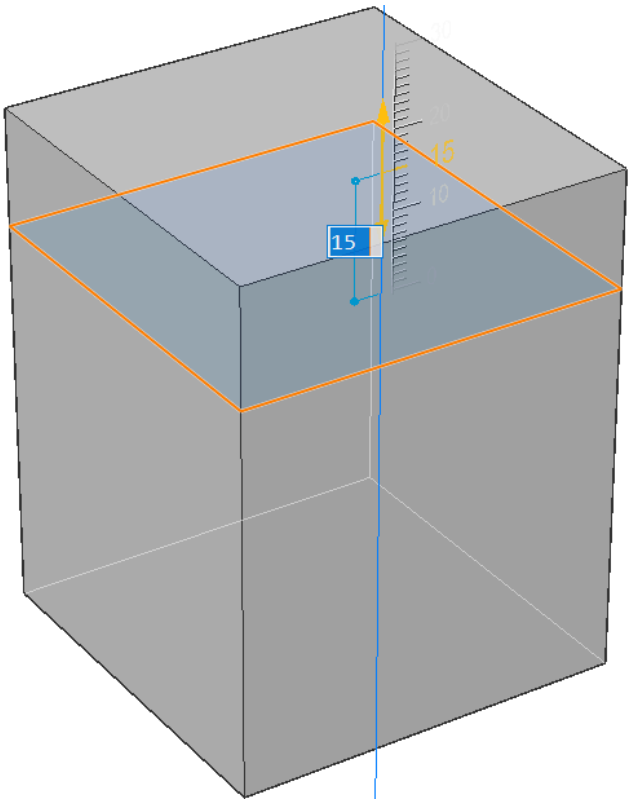
- When you highlight an entity and extrude it to the inward, a volume is subtracted from the main solid.

- 

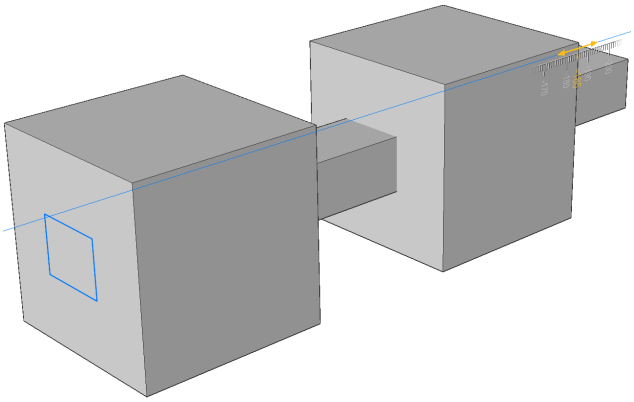
Subtract
The 3D solid is subtracted from each interfering existing solid.



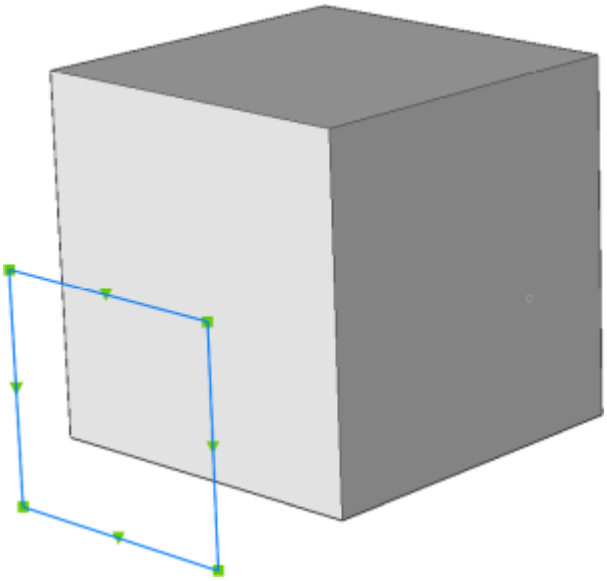
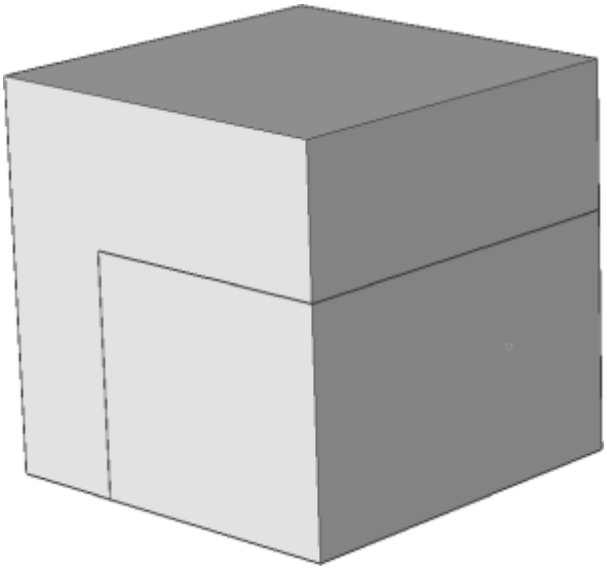
Create
A new volume or surface is created, regardless of the extrusion direction.



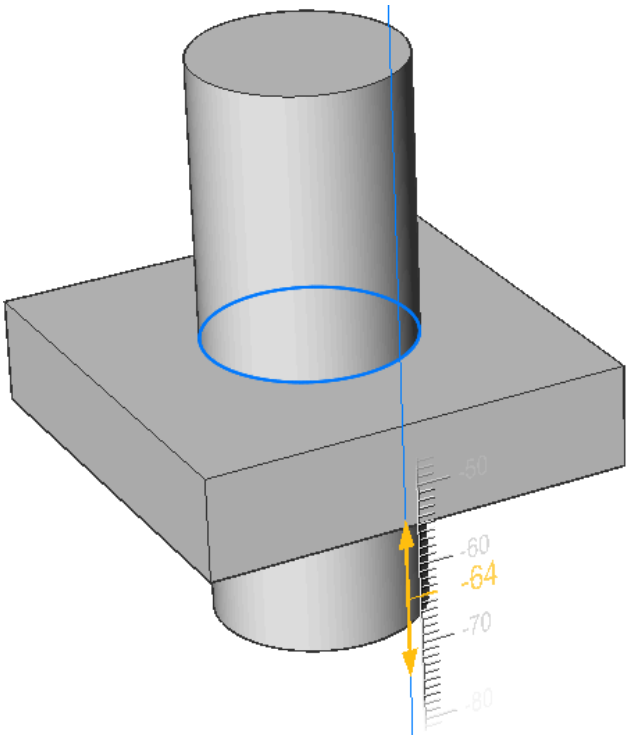
Unite
The new 3D solid is unified with each interfering existing solid.



Slice
Cuts through solids with the extruded surface.

- Select an entity to be extruded.
- 
- The extruded surface cuts through solids.
- 

Both sides
Extrudes symmetrically in both sides.



ALign
Allows you to select or to define an axis.

2Points
Define the extrusion direction by specifying two points.

Object
Allows you to select an axial entity.

Last
Uses the previous axis.

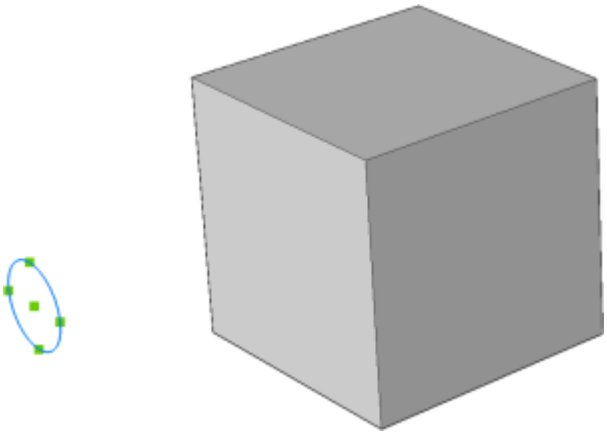
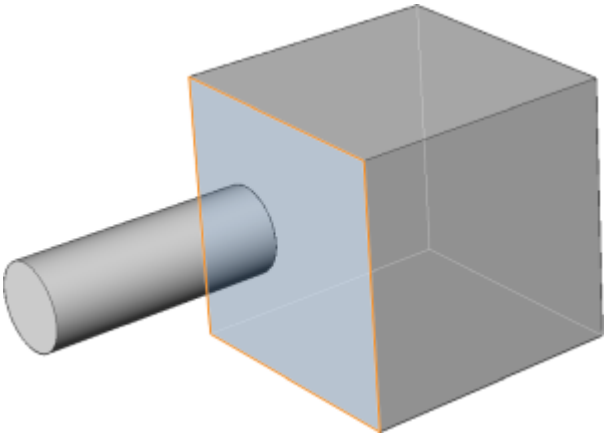
View
Allows you to specify a point on view direction.

Xaxis
Sets the extrusion direction parallel to the X axis of the current UCS.

Yaxis
Sets the extrusion direction parallel to the Y axis of the current UCS.

Zaxis
Sets the extrusion direction parallel to the Z axis of the current UCS.

set Limit
Allows to limit the extrusion by the face of a solid.

- Select an entity to be extruded.
- 
- Select a face as limit of the extrusion.
- 

Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the **Ctrl** key during the dynamic display of the extrusion to cycle the various options.

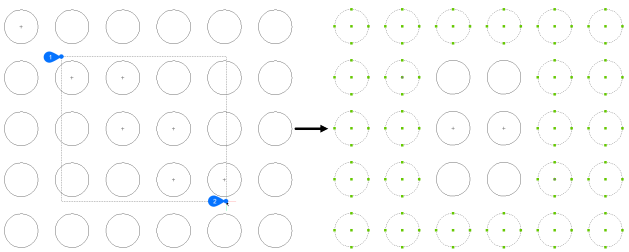
10.43 EXW command (Express Tools)

Selects all entities of your drawing except the ones inside a specified rectangle.

Shape Lite Pro Mechanical BIM

10.43.1 Method

Select the first (1) corner and the second (2) corner to draw a temporary rectangle. All entities in the drawing are selected except the ones inside the rectangle.



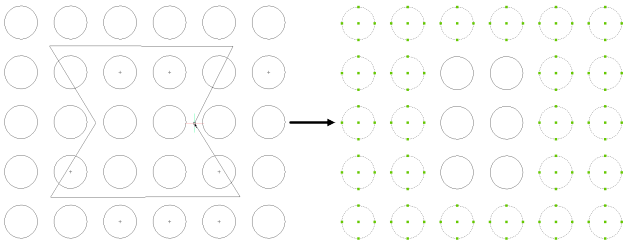
10.44 EXWP command (Express Tools)

Selects all entities in the drawing except the ones inside a specified polygon.

Shape Lite Pro Mechanical BIM

10.44.1 Method

Draw a temporary polygon. All the entities in your drawing will be selected except the ones enclosed by this polygon.



11. F

11.1 FASTSEL command (Express Tools)

Selects all entities that touch the selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

11.1.1 Method

The behavior of the FASTSEL command is controlled by the FSMODE command.

11.2 FBXEXPORT command

Exports 3D entities in the current drawing to FBX format.

Shape Lite Pro Mechanical BIM

Note: If you wish to export 2D entities, you must first give them a thickness.

11.2.1 Method

There are two methods to export to FBX format:

- Export all visible entities.
- Export only the selected entities.

11.2.2 Options within the command

Selected
Select the entities to be exported.

Visible
Exports all visible entities.

Note: This option excludes the entities on frozen or off layers, and those that are not visible in the current viewport.

Select
Prompts to select among entities, lights, cameras and materials.

All
Exports all entity types.

Embedded
Includes the texture files into the FBX file.

Reference to file
Adds a reference link in the FBX file to the location of the texture files.

Copy to file
Copies the texture files to the destination folder, separately from the FBX file.

Note: Texture files are not included in the FBX files when they are very large or when a common set of texture files are used by rendering and animation projects.

Enter path to export fbx file:
Specify the path to the folder in which to place the FBX file, or press Enter key to accept the path provided.

Note: Typing ~ displays the FBX export dialog box, which allows you to select a folder.

11.3 -FBXEXPORT command

Exports 3D entities in the current drawing to FBX format.

Shape Lite Pro Mechanical BIM

See the FBXEXPORT command.

11.4 FIELD command

Opens the **Field** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

11.4.1 Method

Open the **Field** dialog box to create a field in the current drawing.

11.4.2 Options within the command

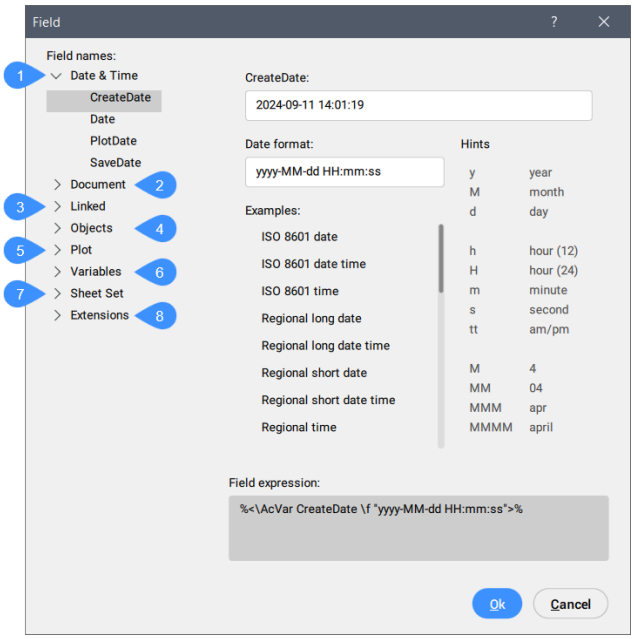
Specify start point
Specifies a point in the current drawing where the field will be inserted.

Height
Specifies the height of the field text.

Justify
Specifies a justification for the field text.

The **Field** dialog box allows you to insert a field in a single or multiline text, table cells, attributes, and so on.

Note: A field is variable text that uses a coding system which the program updates automatically, displaying data as drawing properties (for example a file name or the saved date), entity properties (like length, area, or layer), user-defined properties, plot settings, variables... If a field does not have value, the program displays hyphens (----). If the information referenced by the selected property is not valid, the value for the field is displayed as ####.



Note: Expand a category in **Field Names**, then select a field name.

1 Date & Time

2 Document

3 Linked

4 Objects

5 Plot

6 Variables

7 Sheet Set

8 Extensions

9 Field expression

11.4.3 Date & Time

CreateDate
Displays the last revision date of the drawing.

Date
Displays the current date.

PlotDate
Displays the date when the drawing was last plotted.

SaveDate
Displays the date when the drawing was last saved.

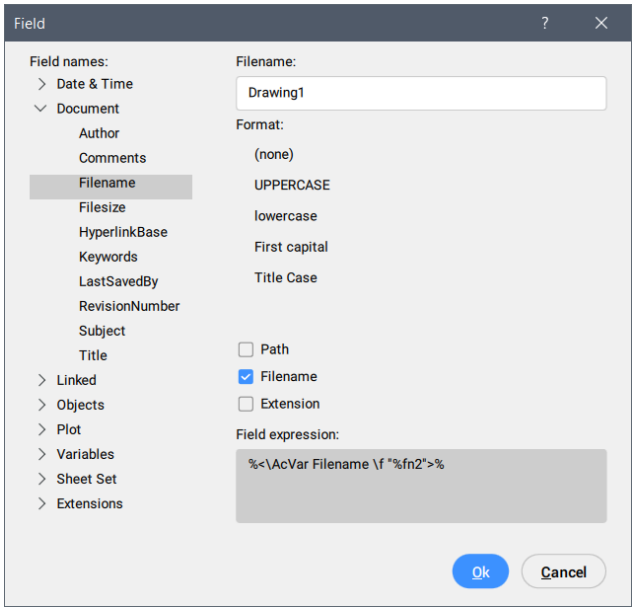
Date format
Select a date format from the list or type a format in this field.

Examples
Displays date format examples.

Hints
Explains the meaning of the 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 surrounded with arrow brackets (<>).

11.4.4 Document



Author
Displays the author of the document.

Comments
Displays the comments in the document.

Filename
Shows the file name. 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 the relative hyperlink in drawings.

Keywords
Displays the keywords for the opened file.

LastSavedBy
Displays the name of the person that last saved the file.

RevisionNumber
Displays the drawing revision number.

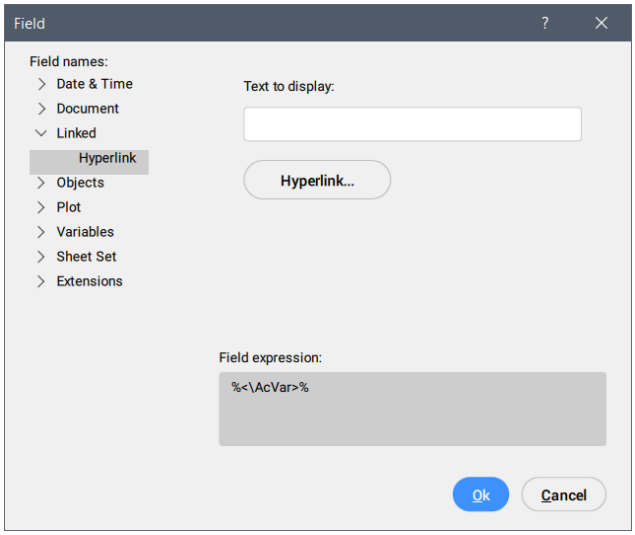
Subject
Displays the subject of the drawing.

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 surrounded with arrow brackets (<>).

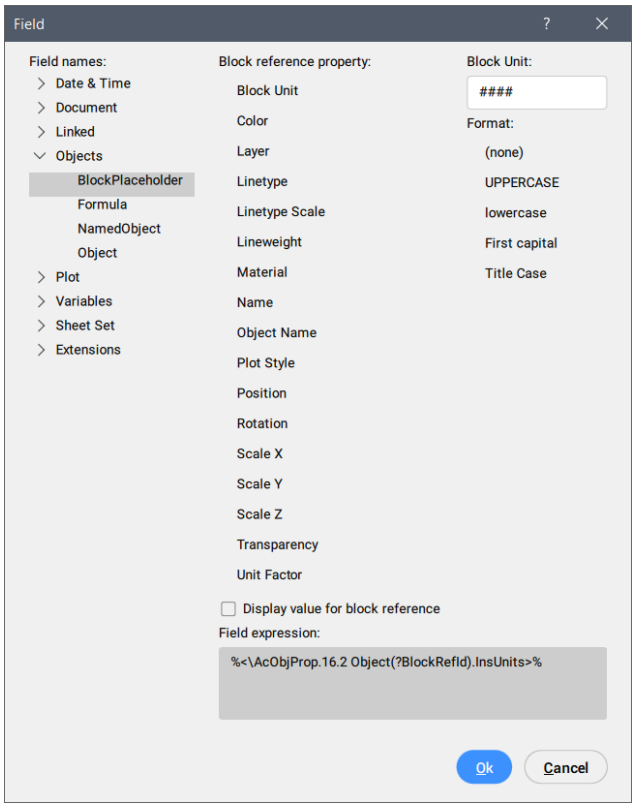
11.4.5 Linked



Text to display
Enter the text to be displayed for this hyperlink.

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

11.4.6 Objects



BlockPlaceholder

See the ATTDEF command to create block property fields in an attribute. When such an attribute is included in a block definition, the field displays the current value of the block property.

Block reference property

Displays the block reference property.

Block name

Displays the name of the property.

Format

Specifies the format for displaying the property.

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 box closes to allow you to select cells in a table in the drawing. Also, you can enter the formula manually, for example, if you know the table handle.

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

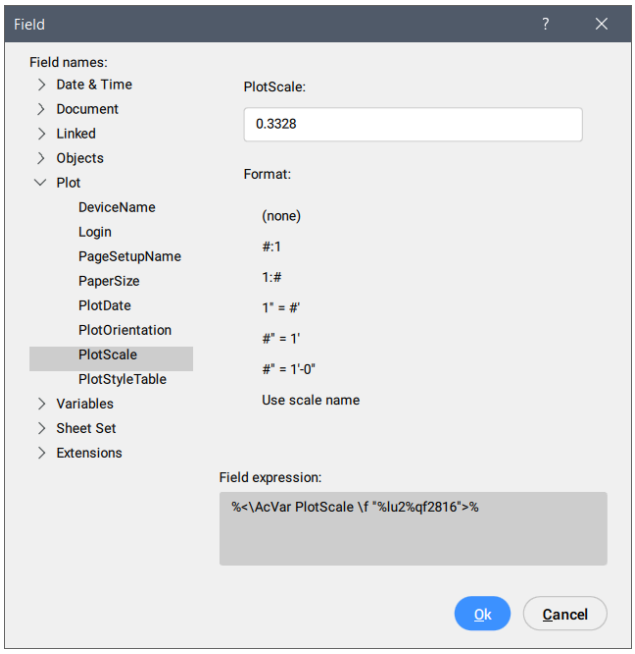
Creates a field that shows a property of a selected entity.

- Click the select button, next to the **Object type** field.

  The **Field** dialog box 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. Custom properties of all parameter types and visibility states for dynamic blocks and block references are also available.
- Choose a format.
- Click the **OK** button to place the field.

  Note: Attributes with fields are updated after REGEN and UPDATEFIELD commands.

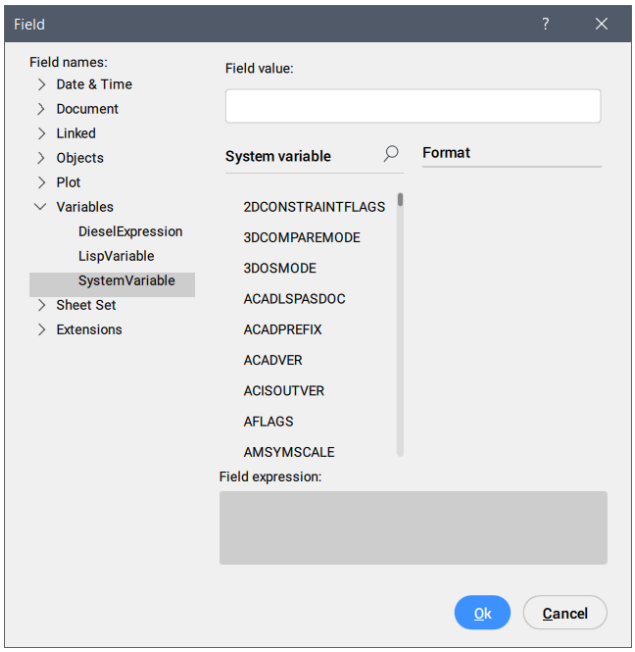
11.4.7 Plot



Plot fields apply to paper space layouts.

Allows you to create fields which show the current print settings of a layout.

11.4.8 Variables



Allows you to create fields which show the current value of **Diesel Expressions**, **Lisp Variables** and **System Variables**.

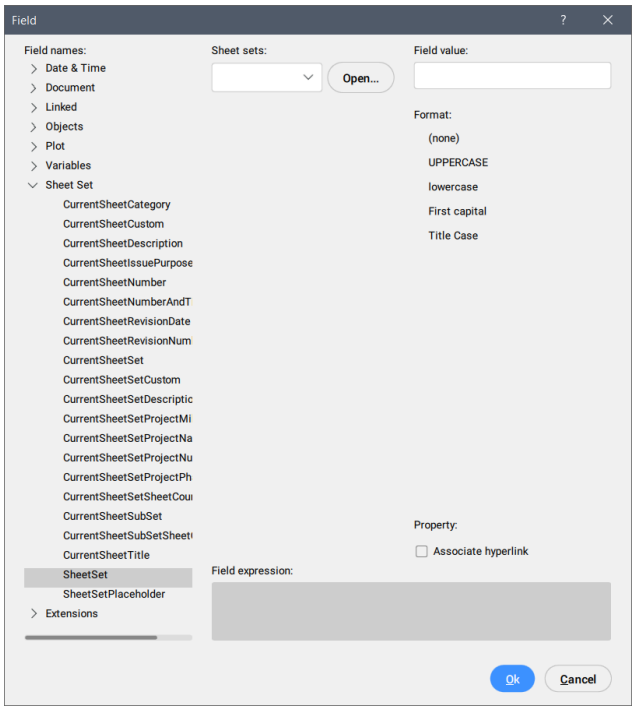
It is possible to search for a specific variable in the **Lisp Variables** and **System Variables** lists using:

- **Key navigation**: press the up/down arrows to navigate in the list.
- **Search bar**: allows you to search for specific variables. The list is shortened to display only the variables that contain the entered sequence of characters.

  Note: To enable the search bar:

  - Click the icon.
  - Press the **Crtl+F** key. The focus needs to be in the variable list. You can use the TAB key to cycle focus.

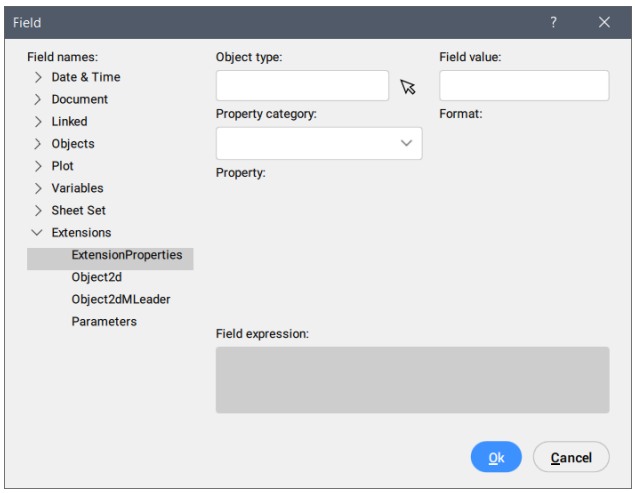
11.4.9 Sheet Set



Allows you to create fields which show default and custom sheet set properties. With the **Sheet Set** property, a sheet navigation tree is opened.

Note: Sheet Set fields are updated with the UPDATEFIELD command.

11.4.10 Extensions



Allows you to create fields that show properties of entities specific to BricsCAD, such as BIM or Mechanical entities, displaying cached values in AutoCAD®.

ExtensionProperties

Allows you to create fields with properties of a selected BIM entity.

Object2d
Allows you to create fields with properties of 3D entities by selecting the associated 2d entity inside BIM section block.

Object2dMLeader
Allows you to create fields inside MLeader with properties of 3D entities by selecting the associated 2d entity inside BIM section block using MLeader's arrow.

Parameters
Allows you to create fields referencing parameters in the drawing.

Note:

- For block parameters, use the FIELD command inside a BEDIT session.
- After changing parameter values (from the **Properties**, **Parameters Manager**, or **Mechanical Browser** panel) use the REGEN, REGENALL, or UPDATEFIELD command to update the value displayed in the created field.

11.4.11 Field expression

Displays the expression of the field. You can learn how fields are structured by reading this code.

11.5 FILEOPEN command

Opens files from the Command line.

Shape Lite Pro Mechanical BIM

11.5.1 Description

Opens drawing (DWG), template (DWT), and interchange (DXF) files from the command line, replacing the current drawing.

11.5.2 Options within the command

Save the changes to drawing file?
Specifies whether the current drawing should be saved or not.

Open Drawing
Type the name of the drawing, including path.

Note: Enter ~ to display the **Open Drawing** dialog box.

11.6 FILES command

Opens the operating system's file manager to access files.

Shape Lite Pro Mechanical BIM

11.7 FILL command

Toggles the FILLMODE system variable.

Shape Lite Pro Mechanical BIM

11.7.1 Description

Toggles the FILLMODE system variable to specify the display of filled 2D entities including polyline, hatch, solid, and trace entities. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'FILLMODE. You must use REGEN or REGENALL to see the change.

- On: turns on the FILLMODE system variable
- Off: turns off the FILLMODE system variable

11.8 FILLET command

Fillets intersections; joins intersecting lines with an arc of any suitable radius.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: F

11.8.1 Description

Sets a radius value for fillet between two entities.

The entities could be:

- Lines, including parallel lines
- All vertices of a single 2D polyline; two polylines cannot be filleted
- Rays, including parallel rays
- Infinite lines, including parallel infinite lines
- Arcs

Note:

- Press and hold the Shift key when select the second entity to create a corner (radius = 0). The selected entities are extended or trimmed at the intersection point. The selected part of the entities is kept.
- If the second entity is parallel to the first entity, the entities are connected by a semicircle. If the parallel entities are not of equal length, then the second entity is extended or trimmed when the **Trim** option in the Command line is set to **Trim** (TRIMMODE system variable = 1).

11.8.2 Options within the command

fillet Settings

Displays the **Chamfer/Fillet** section in the **Settings** dialog box:

Chamfer/Fillet	
Chamfer mode	[0] Distance-Distance
Chamfer first distance	0 mm
Chamfer second distance	0 mm
Chamfer length	0 mm
Chamfer angle	0
1 Fillet radius	10 mm
2 Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

1 **Fillet radius**: sets the radius of the fillet arc.

2 **Trim mode**: toggles whether the entities are trimmed back to meet the fillet radius.

Polyline
Fillets the all vertices of a polyline.

Radius
Toggles edge mode.

Trim
Sets the trim mode for chamfers and fillets:

Trim
Trims or extends the selected entities.

No trim
Creates the chamfer or fillet, but leaves the selected entities unchanged.

Note: No new chamfer or fillet entities will be created when the TRIMMODE and FILLETRAD system variables are set to 0.

Undo
Undoes the last fillet when in **Multiple** mode.

Multiple
Creates additional fillets using the same settings without having to restart the command.

11.9 FIND command

Opens the **Find and Replace** dialog box.

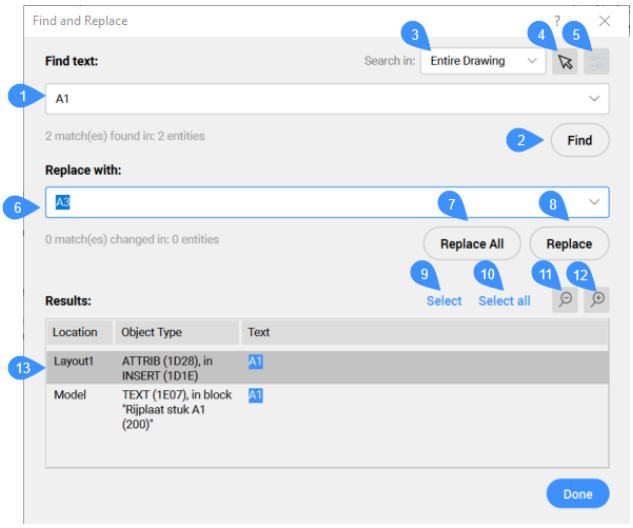
Shape Lite Pro Mechanical BIM

Icon:

11.9.1 Description

Opens the **Find and Replace** dialog box.

The **Find and Replace** dialog box allows you to find and optionally replace strings of text in the current drawing. It searches for text in blocks, attributes, dimensions, and hyperlinks.



1 Find Text

2 Find

3 Search In

4 Select Entities

5 Options

6 Replace With

7 Replace All

8 Replace

9 Select

10 Select all

11 Zoom Out Drawing

12 Zoom In Drawing

13 Search report

11.9.2 Find Text

Specifies the text to find.

11.9.3 Find

Finds the next occurrence of the text string.

Shortcut Keystroke: Crtl+F.

11.9.4 Search In

Specifies the range of search:

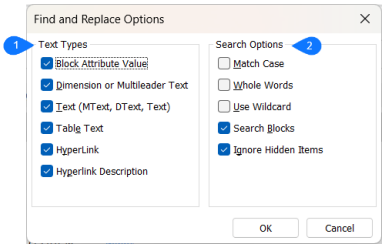
- **Entire Drawing**: searches all text in the drawing.
- **Current Layout**: only searches in the current Layout.
- **Current Selection**: only searches in the current selection that can be made using button no. 5.

11.9.5 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.

11.9.6 Options

Opens the **Find and Replace Options** dialog box that allows you to specify the types of text entities to search.



Text Types

Specifies the types of text to include in the search:

- Block Attribute Value
- Dimension or Multileader Text
- Text
- Table Text
- HyperLink
- Hyperlink Description

Search Options

Specifies the search options:

- **Match Case**: only reports if string matches the use of capital letters.
- **Whole Words**: only reports if whole word matches the entered text string. If a long word contains the entered string, this will not be reported in the results.
- **Use Wildcard**: filters the search using wildcard characters as comparison criteria.
- **Search Blocks**: includes blocks in the search.
- **Ignore Hidden Items**: does not include hidden items in the search.

11.9.7 Replace With

Specifies the text that will replace the found text. Do not fill in this field when you're only searching for text.

11.9.8 Replace All

Replaces all occurrences of the text string with the replacement string.

Shortcut Keystroke: Ctrl+A.

11.9.9 Replace

Replaces the found text string with the replacement string.

Shortcut Keystroke: Ctrl+R.

11.9.10 Select

Creates a selection set from the selected entities in the list and closes the dialog box.

11.9.11 Select All

Selects all occurrences of the text string.

11.9.12 Zoom Out Drawing

Zooms out from the center of the window with a factor of ½.

11.9.13 Zoom In Drawing

Zooms in on the center of the window with a factor of 2.

11.9.14 Search report

Displays the number of matches found in the selected entities, and the number of matches and entities changed after replacement. The search result can be sorted by each column.

11.10 FINDOUTLIERS command

Finds any outlier outside of the valid area.

Shape Lite Pro Mechanical BIM

Icon: 

11.10.1 Description

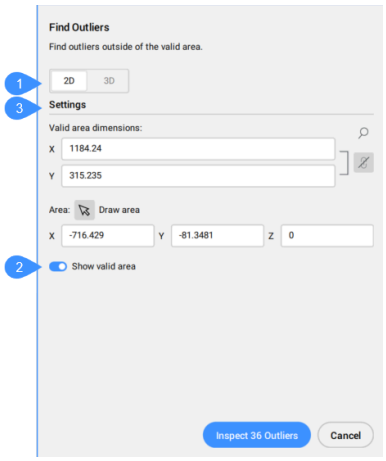
Finds entities positioned at extreme coordinates outside a specified area. These entities are often hard to detect and can make it difficult to manipulate the view because it depends on the total extent of the drawing.

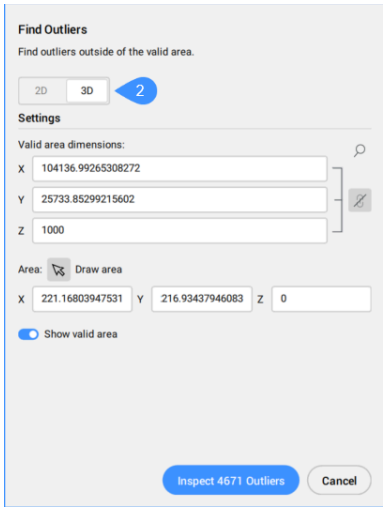
Note: Invisible entities and entities on invisible or locked layers are detected and temporarily highlighted but cannot be moved or deleted.

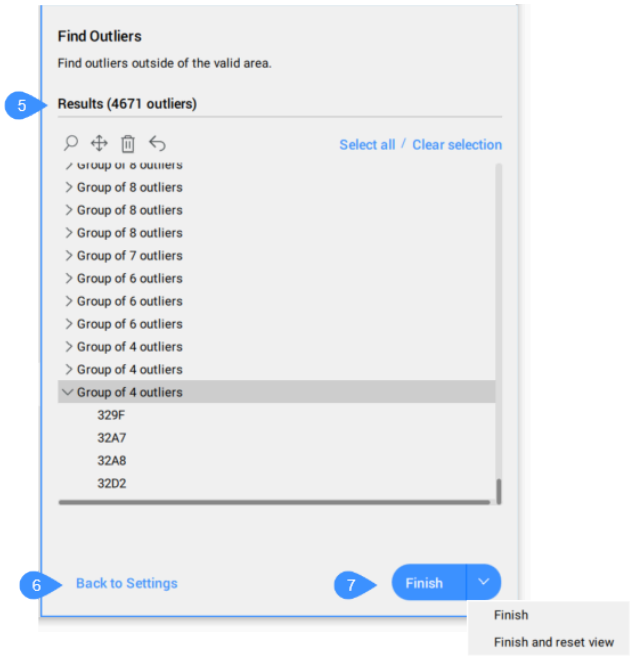
Note: Entities on a frozen layer are detected but cannot be temporarily highlighted, moved, or deleted.

11.10.2 Method

The command opens the **Find Outliers** command panel.


Find Outliers
Find outliers outside of the valid area.
1
2D
3D
3
Settings
Valid area dimensions:
X 1184.24
Y 315.235
Area: Draw area
X -716.429
Y -81.3481
Z 0
2
Show valid area
Inspect 36 Outliers
Cancel


Find Outliers
Find outliers outside of the valid area.
2D
3D
2
Settings
Valid area dimensions:
X 104136.99265308272
Y 25733.85299215602
Z 1000
Area: Draw area
X 221.16803947531
Y 216.93437946083
Z 0
Show valid area
Inspect 4671 Outliers
Cancel


Find Outliers
Find outliers outside of the valid area.
5
Results (4671 outliers)
Select all / Clear selection
Group of 8 outliers
Group of 8 outliers
Group of 8 outliers
Group of 7 outliers
Group of 6 outliers
Group of 6 outliers
Group of 6 outliers
Group of 4 outliers
Group of 4 outliers
Group of 4 outliers
329F
32A7
32A8
32D2
6
Back to Settings
7
Finish
Finish
Finish and reset view

1 2D mode

2 3D mode

3 Settings

4 Show valid area

5 Inspect Outliers

6 Results

7 Back to Settings

8 Finish

Note: The options within the Command line reflect the options within the command context panel.

11.10.3 2D mode

When this mode is active, the default area is simplified to a 2D rectangle, defined by the X and Y dimensions.

Note: The default value of the Z dimension is zero, but you can change it anytime.

11.10.4 3D mode

When this mode is active, the valid area is a 3D box, defined by the X, Y, and Z dimensions.

11.10.5 Settings

Valid area dimensions
Sets the valid area of the drawing. The entities outside this area are considered to be at extreme coordinates and are listed in the **Results** section of the **Find Outliers** command panel.

Note: In the Command line, the **change Dimension** option allows you to set the valid area dimensions.

The default value for the X, Y and Z dimensions is 100000.

Note:

- Click the **Sync all dimensions** button ( ) to sync or unsync all dimension.
  - When the **Sync all dimensions** button is On and you modify one dimension, the other dimensions change to the same value.
  - If the dimensions have different values (the dimensions are unsynced) and you turn On the **Sync all dimensions** button, the Y (for 2D mode), respectively the Y and Z (for 3D mode) values change to the same value as X.
- The valid area is graphically displayed in the model space by a transparent green box.
- The unit used depends on the template you are working in. For example, if you are working in the Default-mm template, the unit for the valid area is mm.
- In the Command line, the **Keep all dimension identical** option defines if all dimensions will be modified at once.
- Click the magnifying glass icon ( ) to zoom to extents.

Area
Sets the area directly on the drawing Model Space, by picking the points for each coordinate.

Note: In the Command line, the **draw new Area** option allows you to define the area by picking points for each coordinate.

11.10.6 Show valid area

Enables/Disables the display of the valid area.

Note: In the Command line, the **change Show valid area** option allows you to Enables/Disables the display of the valid area.

11.10.7 Inspect Outliers

Displays the **Results** section.

11.10.8 Results

Displays the outliers found in the drawing. You can select the outlier by clicking it. The entity is then selected in the drawing. After selecting an outlier, you can press the magnifying glass icon () to zoom in, the move icon () to **Move to valid region origin** or in any other place, or the trashcan icon () to delete it. Also, you can undo the move or delete actions by pressing the undo icon ().

The same options are available in the context menu opened by right clicking the selected outliers.

Note: To temporarily highlight invisible entities and entities on invisible or locked layers, double-click on them.

Note: After moving an outlier to the valid region, the list of outliers will be updated accordingly.

Outliers that lie close to each other are clustered as a group. When selecting a group of outliers, all the elements in the group are selected at once to zoom in on them, move, or delete them. If there is only one group present, the entities are listed individually, and not clustered in a group.

Note: You cannot modify the selected entity while the command is running.

When selecting outliers, a widget appears in the drawing area. The widget gives you control to zoom in () to the entity attached to it or zoom out () when clicking it again.

Note:

- Click the widget of a cluster to zoom to the extents of the cluster. The zoom widget of the cluster disappears and individual widgets get displayed for the individual outliers in the cluster.
- If you select more than 1 cluster in the command panel list (for example: 2 clusters, or 1 cluster and 4 individual outliers) and use the action menu to zoom to them, widgets are only displayed for the entire clusters, not on the level of the individual outliers in the cluster.
- If you select one cluster in the command panel list and use the action menu to zoom to them, all individual outliers of the cluster have a widget.
- Selected visible outliers that do not belong to a cluster have a widget.

11.10.9 Back to Settings

Goes back to the **Settings** section.

Note: In the Command line, the **Back** option allows you to go back to **Inspect Outliers** section.

11.10.10 Finish

Closes the **Find Outliers** command panel and keeps the selected outliers.

Finish and reset view
Closes the **Find Outliers** command panel and restores the view (zoom level, camera position).

Note: In the Command line, the option is **finish and Reset view**.

11.11 FITARC command

Fits an arc or a circle to entities.

Shape Lite Pro Mechanical BIM

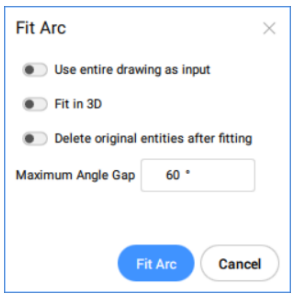
11.11.1 Description

Draws an arc or circle fitted to one or more entities. The command minimizes accumulated perpendicular distances to the points of the selected entity(ies).

A realistic use case is a production company of advertisement cardboard. Typically, the 2D CAD files to pass to the cutting machine are imported from other design software, where polylines and splines are the typical 2D curve entity types. Take the example where the machine has to cut a half-circle in the cardboard, and the half-circle is represented by a polyline with a lot of small subsequent linear segments. When passing this drawing to the cutting machine, the machine will make a sequence of many of small cuts. After each cut, the machine has to stop, change orientation and restart. For each stop-change-restart action, there is a chance for errors. This would not only lead to a much longer cutting time, also the resulting cut will be suboptimal and look rough. In this case, FITARC could be used to turn this polyline into an ARC, leading to one fast and smooth cut operation.

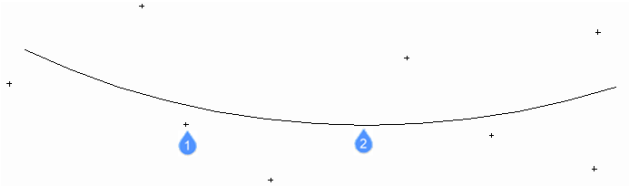
11.11.2 Methods

This command opens the **Fit Arc** command panel.



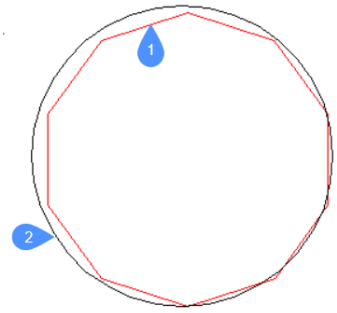
The options within the FITARC command are the same as the options in the **Fit Arc** command panel.

The entities and the results of this command are shown in the examples below:



1 Points used by the command to fit the arc.

2 Arc fitted to the points.



1 Polyline used by the command to fit the arc.

2 Arc fitted to the polyline.

11.11.3 Options within the command

use Entire drawing

Use all entities in the drawing as input.

Fit in 3d

This option applies when one or more input entities do not lie in the XY-plane.

If checked, the resulting arc or circle is created in 3D space, otherwise the input entities are projected onto the XY-plane first.

Note: The XY plane used for projection is the one of the current UCS (User Coordinate System). This allows the user to make the fit in any plane.

Delete original entities after fitting

When this option is set to **Yes**, the initial entities are deleted.

maximum Gap angle

Defines the threshold value for the maximum angle gap between the characteristic points of the input entities. If the maximum gap angle is larger than the threshold value, an arc is created. Otherwise, a circle is created.

selection options (?)

Allows to choose a selection method. See the SELECT command.

Note: The options **Use entire drawing**, **Fit in 3d**, and **Delete original entities after fitting** can be set using the system variable FITLINEFITARCMODE. The option **Maximum gap angle** can be set using the FITARCMAXGAP system variable.

11.12 FITLINE command

Fits a line to entities.

Shape Lite Pro Mechanical BIM

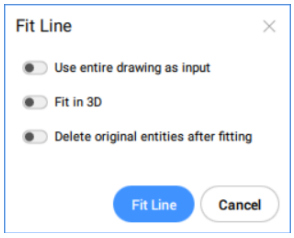
11.12.1 Description

Draws a line fitted to one or more entities. This command minimizes accumulated perpendicular distances to the points of the selected entities.

The FITLINE command works on any input entity type, so can be applied in various real life use cases. You can use it to draw a line which optimally matches a set of scanned points in the XY-plane. You can use FITLINE to find the optimal line that fits a multi-segment polyline or spline curve that goes up and down.

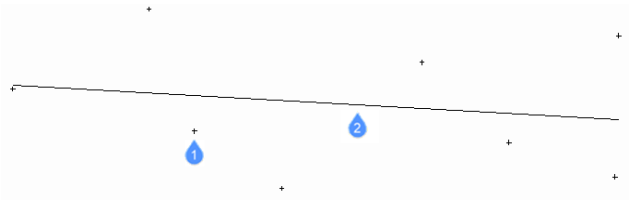
11.12.2 Methods

The command opens the **Fit Line** command panel.



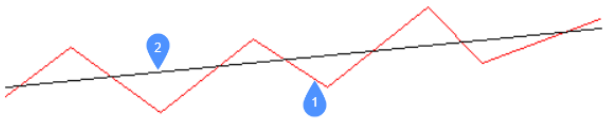
The options in the FITLINE command are the same as the options in the **Fit Line** command panel.

The entities and the results of this command are shown in the examples below:



1 Points used by the command to fit the line.

2 Line fitted to the points.



1 Polyline used by the command to fit the line.

2 Line fitted to the polyline.



1 Spline used by the command to fit the line.

2 Line fitted to the spline.

11.12.3 Options within the command

use Entire drawing
Use all entities in the drawing as input.

Fit in 3d
This option applies when one or more input entities do not lie in the XY-plane.
If checked, the resulting line is created in 3D space, otherwise the input entities are projected onto the XY-plane first.

Note: The XY plane used for projection is the one of the current UCS (User Coordinate System). This allows the user to make the fit in any plane.

Delete original entities after fitting
When this option is set to Yes, the initial entities are deleted.

selection options (?)
Allows to choose a selection method. See the SELECT command.

Note: The options **Use entire drawing**, **Fit in 3d**, and **Delete original entities after fitting** can be set using the FITLINEFITARCMODE system variable.

11.13 FITPOLYLINE command

Fits polylines to entities.

Shape Lite Pro Mechanical BIM

Icon:



11.13.1 Description

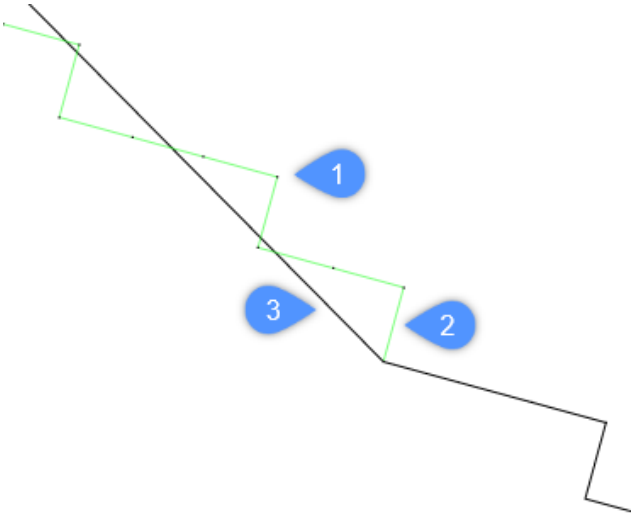
Draws one or multiple polyline entities, fit to the characteristic points of the input entities. The command minimizes accumulated perpendicular distances to the points of the selected entities. The command works with every type of entity, as long as it has characteristic points.

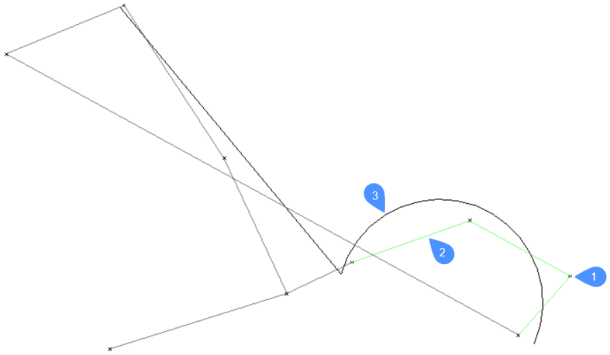
11.13.2 Method

There are two ways to define the input to the FITPOLYLINE command:

- Select fitting entities: choose the entities to which the polyline should be fit.
- Use entire drawing: select all non-frozen entities in the current viewport.

Pick a point for Start of polyline. The guideline is used as a visual aid for the user, helping to pick the envisioned polyline vertices.





1 The points used by the command to fit the polyline.

2 The suggested guideline.

3 The polyline.

11.13.3 Options within the command

use Entire drawing
Selects all non-frozen entities in the current viewport.

Select fitting entities
Allows you to select the entities to which the polyline should be fit.

Note:

- If you select a point cloud and a section, the FITPOLYLINE command automatically looks for points inside that section and works with these points.
- If you select a point cloud but not a section, the application asks to select a section.
- If you select a section but not a point cloud, the application asks to select a point cloud.

Selection contained multiple pointclouds, select one
Allows you to select a specific point cloud.

Note: This option becomes available when there is more than one point cloud in the selection set.

Selection contained multiple sections, select one
Allows you to select a specific section.

Note: This option becomes available when there is more than one section in the selection set.

change section crop mOde
Allows you to choose between **points Inside section** and **points Below section**.

Start the polyline
Choose a point and start the polyline.

fit Lines
Fits a linear polyline segment to the points between the last fixed polyline vertex and the mouse position.

Note: See the FITLINE command.

fit Arcs
Fits an arc polyline segment between the points between the last fixed polyline vertex and the mouse position.

Note: See the FITARC command.

Best fit
The algorithm automatically decides whether a linear or an arc polyline segment is best suited to fit the points between the last fixed polyline vertex and the mouse position.

Note: If the HOTKEYASSISTANT system variable is set ON, press the **Ctrl** key to cycle between the fit options.

Undo
Undoes the last action.

Close polyline
Closes the polyline.

start New polyline
Starts a new polyline.

Note: Press the **Esc** key once to end the current polyline and start a new polyline.

Note: Press the **Esc** key twice to finish the command.

Finish
Finishes the command.

11.14 FLATSHOT command

Creates a flattened, hidden line representation of a 3D model.

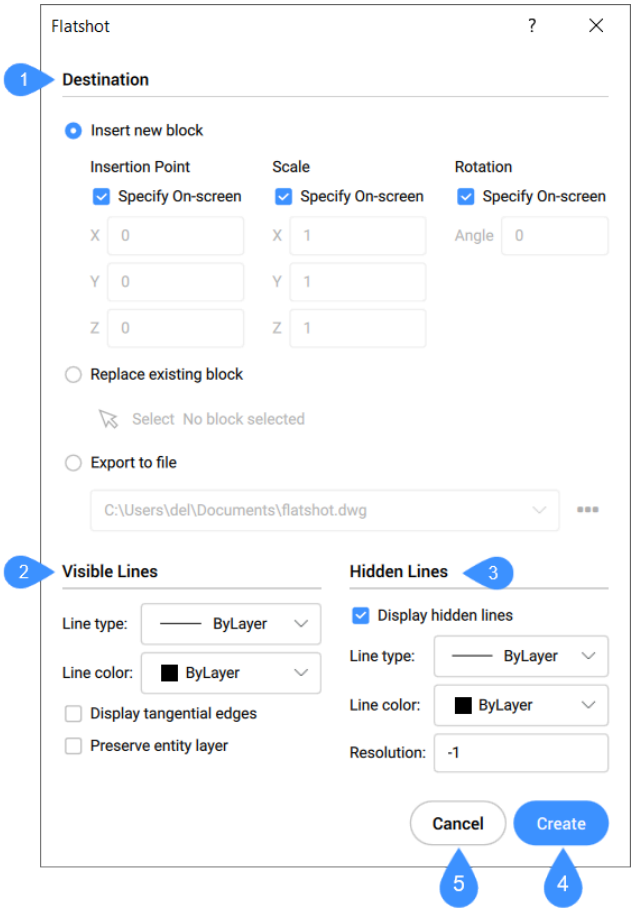
Shape Lite Pro Mechanical BIM

Icon: 

11.14.1 Description

Opens the **Flatshot** dialog box.

The **Flatshot** dialog box allows you to create a flattened, hidden line representation of 3D solids, and 3D surfaces as a block or a new drawing.



1 Destination

2 Visible lines

3 Hidden lines

4 Create

5 Cancel

11.14.2 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:

- Insertion point
- Scale
- Rotation

Replace existing block
Replaces a block that is already inserted in the drawing. By choosing this option, you can further select a block from the current drawing.

Export to file
Saves the flatshot block as a DWG file through the **Select Export File** dialog box.

11.14.3 Visible lines

Sets the linetype and color for the visible lines. Choose the linetype and color from the drop-down lists.

Note: You can load a new linetype into the drawing through the **Load Linetypes** dialog box by selecting **Load...**. You can select a new line color through the **Color** dialog box by selecting **Select color...**.

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

Preserve entity layer
When activated, the layers of entities in the block are maintained.

11.14.4 Hidden lines

Toggles the display of hidden lines and sets their properties.

Display hidden lines
Toggles the display of hidden lines.

Resolution
Specifies the short entity that will be represented by a hidden line. Enter a negative value so the program will determine this automatically. The range is from 0.01 to 0.000000000001.

11.14.5 Create

Creates the Flatshot.

Note: The corresponding keyboard shortcut for this button is **ALT + R**.

11.14.6 Cancel

Cancels the Flatshot creation.

Note: The corresponding keyboard shortcut for this button is **ALT + C**.

11.15 FLATTEN command

Flattens 2D and 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

11.15.1 Description

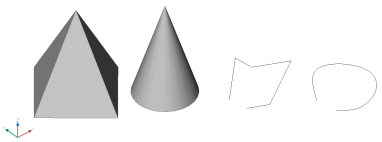
Flattens 2D and 3D entities by projecting them onto the XY-plane of the current view.

Note: The PERSPECTIVE system variable must be set to 0.

11.15.2 Method

The entities will be projected onto the XY plane of the current view. In top view, the entities will be projected onto the XY-plane of the current coordinate system (WCS or UCS).

Entities before FLATTEN command:



Entities after FLATTEN command:



- Pyramid converted to a square with its diagonals.
- Cone becomes a circle with a point in the center.
- 3D polyline and spline are converted to 2D polylines.

11.15.3 Options within the command

new ucs ELevation

Specifies the new elevation for the resulting 2D entities.

Note: The elevation and thickness of the 2D entities can be changed with the ELEV command.

allow EXplode

Explodes compounded entities, such as 3D solids.

11.16 FLIPLINE command

Creates a line, used to flip the entities of a block.

Shape Lite Pro Mechanical BIM

Icon:

Alias: PFLIP, PARAMETRICFLIP

11.16.1 Method

This command is used with 2D parametric blocks.

The FLIPLINE command allows you to draw a line used as a reference axis for flipping the selected entities of the block.

The command will automatically create a special type of parameter and will associate it to the flip line entity. This parameter has two possible values: **Not Flipped** and **Flipped**.

Note:

- Creating a flip line is equivalent to drawing a line using the LINE command. For more info about using the FLIPLINE command, see the article **2D parametric blocks**.
- Creating a flip line on an entity already affected by an existing flip line can lead to unexpected behavior.

11.16.2 Options within the command

Select entities to flip
Lets you select which entities should be flipped. The constraints affecting the selected entities will not be flipped.

flip Everything
Selects all entities in the drawing for flipping. The constraints group will be flipped as well.

To flip the block, change the value of the flip parameter:

- select the parametric block and click the flip parameter's grip.
- access this parameter in the **Parameters Manager** panel.
- select the block and open the **Properties** panel, in the **Parameters** section.

When the flip parameter is set to **Not Flipped**, the block is in its default configuration, as it was drawn.
When the flip parameter value is changed to **Flipped**, the block is flipped around the flip line.

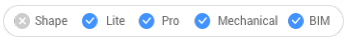
Note: Use the FLIPLINEEDIT command to edit a flip operation.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.
- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

11.17 FLIPLINEEDIT command

Edits parametric flip operations.



Alias: PFLIPEDIT, PARAMETRICFLIPEDIT

11.17.1 Method

The command lets you edit an existing flip operation.

Enter the name of the desired flip operation, then select the parametric operation data to be edited.

Note: To dynamically edit a flip operation within a parametric block, select the block and click the flip operation's grip point.

11.17.2 Options within the command

Selection
Lets you edit the elements selection of the specified operation.

Line
Lets you define a new flip line by selecting its start and end points.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

11.18 FROM command

Allows you to enter relative coordinates with respect to a reference point whenever a command prompts you to enter a point.

Shape Lite Pro Mechanical BIM

Icon: 

Note: From is a command modifier, not a command, and so it is entered only during a drawing or editing command.

Note: This command is useful for starting entities offset from another entity, such as inserting a door relative to the end of a wall.

11.18.1 Options within the command

Base point
Pick a point or enter coordinates to specify the base point.

Offset or regular point
Enter the offset by specifying relative coordinates, using the @x,y,z format

11.19 FS command (Express Tools)

Selects all entities that touch the selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

11.19.1 Method

The behavior of the FS command is controlled by the FSMODE command.

11.20 FSMODE command (Express Tools)

Controls the chain selection during the FS or FASTSEL commands.

Shape Lite Pro Mechanical BIM

11.20.1 Options within the command

OFF

Only the entities that touch the selected entity are selected.

ON

All entities that touch the selected entities and the ones that touch those are selected. The selection goes on until all the connecting entities are selected.

12. G

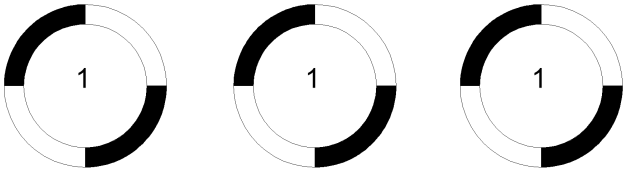
12.1 GATTE command (Express Tools)

Changes the attribute values for all instances of a specified block.

Shape Lite Pro Mechanical BIM

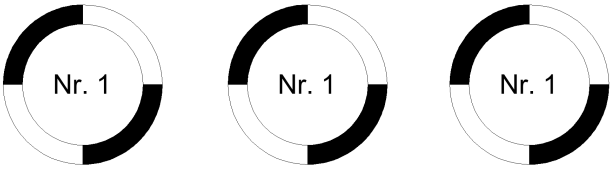
12.1.1 Method

1 Enter a block name or select an attribute.

1 

2 Enter the new text.

3 Reports the number of blocks found and changes the attribute value.

4 

12.1.2 Options within the command

Yes
Automatically changes all attribute values.

No
Allows you to select the attributes you want to change.

12.2 GCCOINCIDENT command

Creates a coincident geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.2.1 Description

Creates a coincident constraint to ensure a point on an entity remains coincident with a specified point or entity.

12.2.2 Methods

This command has 3 methods to begin creating a coincident constraint:

- Select first point
- Entity

- Autoconstrain

12.2.3 Options within the command

Select first point
Allows you to begin creating a coincident constraint by specifying a point on an entity.

Select second point
Specifies a point on an entity to make coincident with the first point. The point on the first entity maintains its position while the second entity adjusts, as necessary, to become coincident.

Entity
Allows you to begin creating a coincident constraint by selecting an entity.

Select a point
Specifies a point on an entity to make coincident with the first entity. The first entity maintains its position while the point on the second entity adjusts, as necessary, to become coincident.

Multiple
Creates multiple coincident constraints until you press Enter to end the command.

Autoconstrain
Creates coincident constraints by selecting all entities to which you want to apply relevant coincident constraints.

12.3 GCCOLLINEAR command

Creates a collinear geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.3.1 Description

Creates a collinear constraint to ensure two or more linear entities remain collinear.

12.3.2 Method

This command has 2 methods to begin creating a collinear constraint:

- Select first entity
- Multiple

12.3.3 Options within the command

Select first entity
Allows you to begin creating a collinear constraint by selecting a linear entity.

Select second entity
Allows you to select a linear entity to make collinear to the first one. The first entity maintains its position while the second entity adjusts, as necessary, to become collinear.

Multiple
Creates multiple collinear constraints.

Select entity to make collinear to the first
Allows you to select a linear entity to make collinear to the first one. You can continue adding collinear constraints to other entities until you press **Enter** key to end the command. The first entity maintains its position while subsequent entities adjust, as necessary, to become collinear.

12.4 GCCONCENTRIC command

Creates a concentric geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.4.1 Description

Creates a concentric constraint to ensure two circular or elliptical entities remain concentric.

12.4.2 Method

This command has 1 method to begin creating a concentric constraint:

- Select first entity

12.4.3 Options within the command

Select first entity
Allows you to begin creating a concentric constraint by selecting a circular or elliptical entity.

Select second entity
Allows you to select a circular or elliptical entity to make concentric with the first entity. The first entity maintains its position while the second entity moves, as necessary, to become concentric.

12.5 GCENTER command

Toggles the **Geometric** center entity snap.

Shape Lite Pro Mechanical BIM

Icon:

12.5.1 Description

Toggles the **Geometric** entity snap to enable or disable snapping to geometric center. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

12.6 GCEQUAL command

Creates an equal geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.6.1 Description

Creates an equal constraint to ensure circular entities maintain equal radii or linear entities maintain equal lengths.

12.6.2 Method

This command has 2 methods to begin creating an equal constraint:

- Select first entity
- Multiple

12.6.3 Options within the command

Select first entity
Allows you to begin creating an equal constraint by selecting an arc, circle, line, or polyline segment.

Select second entity
Allows you to select a similar entity to make equal to the first. The first entity maintains its radius or length while the radius or length of the second entity updates to match the first entity.

Multiple
Creates multiple equal constraints.

Select entity to make equal to the first
Allows you to select a similar entity to make equal to the first. Continue selecting similar entities until you press Enter to end the command. The first entity maintains its radius or length while the radius or length of all subsequent entities update to match the first entity.

12.7 GCFIX command

Creates a fixed geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.7.1 Description

Creates a fixed constraint on a point or entity to maintain its position in the drawing.

12.7.2 Method

This command has 2 methods to create a fixed constraint:

- Select point
- Entity

12.7.3 Options within the command

Select point
Creates a fixed constraint by specifying a point on an entity. The point maintains its position while other geometry moves, as necessary, to maintain other constraints.

Entity
Creates a fixed constraint by selecting an entity. The entity maintains its position while other geometry moves, as necessary, to maintain other constraints.

12.8 GCHORIZONTAL command

Creates a horizontal geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.8.1 Description

Creates a horizontal constraint to ensure a linear entity or pair of points remains parallel to the x-axis.

12.8.2 Method

This command has 2 methods to begin creating a horizontal constraint:

- Select an entity
- 2 Points

12.8.3 Options within the command

Select an entity
Creates a horizontal constraint by selecting a linear entity to make parallel to the x-axis.

2Points
Creates a horizontal constraint by specifying two points.

Select first point
Allows you to begin creating a horizontal constraint by specifying the first point to make parallel to the x-axis.

Select second point
Allows you to specify the second point to make horizontal to the x-axis. The first point maintains its position while the second point moves to become horizontal with the first point.

12.9 GCPARALLEL command

Creates a parallel geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.9.1 Description

Creates a parallel constraint to ensure linear entities remain parallel to each other.

12.9.2 Method

This command has 1 method to create a parallel constraint:

- Select first entity

12.9.3 Options within the command

Select First Entity
Allows you to begin creating a parallel constraint by selecting a linear entity.

Select Second Entity
Allows you to select a second linear entity to make parallel to the first. The first entity maintains its position while the second entity moves, as necessary, to become parallel to the first entity.

12.10 GCPERPENDICULAR command

Creates a perpendicular geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.10.1 Description

Creates a perpendicular constraint to ensure linear entities remain perpendicular to each other.

12.10.2 Method

This command has 1 method to begin creating a perpendicular constraint:

- Select first entity

12.10.3 Options within the command

Select First Entity
Allows you to begin creating a perpendicular constraint by selecting a linear entity.

Select Second Entity
Allows you to select a second linear entity to make perpendicular to the first. The first entity maintains its position while the second entity moves, as necessary, to become perpendicular to the first entity.

12.11 GCSMOOTH command

Creates a smooth geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.11.1 Description

Creates a smooth constraint to ensure two splines maintain fluid geometric continuity between them.

12.11.2 Method

This command has 1 method to begin creating a smooth constraint:

- Select first spline entity

12.11.3 Options within the command

Select first spline curve
Allows you to begin creating a smooth constraint by selecting a spline curve.

Select second curve
Allows you to select a second spline entity. The first spline maintains its position while the second spline stretches, as necessary, to smoothly connect to the first spline.

12.12 GCSYMMETRIC command

Constrains a symmetric geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.12.1 Description

Creates a symmetric constraint to ensure two entities remain symmetric with respect to a selected line.

12.12.2 Method

There are two methods to begin creating a symmetric constraint:

- Select first entity
- 2Points

12.12.3 Options within the command

Select first entity
Allows you to begin creating a symmetric constraint by selecting a 2D entity.

Select second entity
Allows you to select an entity to make symmetric to the first entity.

Select symmetry line
Allows you to select a line to act as a mirror line between the two entities. The first entity maintains its position while the second entity adjusts, as necessary, to become symmetric about that line.

2Points
Creates a symmetric constraint by selecting two valid points on a 2D entity.

12.13 GCTANGENT command

Creates a tangent geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.13.1 Description

Creates a tangent constraint to ensure curved entities remain tangent to another curved or linear entity.

12.13.2 Options within the command

Select first entity
Allows you to begin creating a tangent constraint by selecting a linear or curved entity.

Select second entity
Allows you to select a second entity to make tangent to the first.
The first entity maintains its position while the second entity moves, as necessary, to become tangent to the first entity.
At least one curved entity must be selected per tangent constraint.

12.14 GCVERTICAL command

Creates a vertical geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.14.1 Description

Creates a vertical constraint to ensure a linear entity or pair of points remains parallel to the y-axis.

12.14.2 Method

This command has 2 methods to begin creating a vertical constraint.

- Select an entity
- 2 Points

12.14.3 Options within the command

Select an entity
Creates a vertical constraint by selecting a linear entity to make parallel to the y-axis.

2Points
Creates a vertical constraint by specifying two points.

Select first point
Allows you to specify the first point to make parallel to the y-axis.

Select second point
Allows you to specify the second point to make horizontal to the y-axis.
The first point maintains its position while the second point moves to become vertical with the first point.

12.15 GENERATEBOUNDARY command

Generates a closed polyline or spline around the boundary of a flat, closed area.

Shape Lite Pro Mechanical BIM

Icon:

12.15.1 Description

Generates a closed polyline or spline around the boundary of a planar closed area, a planar face of a 3D solid or around the area of a hatch pattern.

12.15.2 Method

Select a planar enclosed point to generate a boundary by picking a point inside one of the following entities:

- Closed planar area
- Planar face on a 3D solid; to select the face of a 3D solid hold down the Ctrl key.
- Hatch pattern

The resulting boundary are:

- Polyline - when the boundary is polygonal (has corners, like a square)
- Spline - when the boundary is curved, like a spline.

12.16 GEOGRAPHICLOCATION command

Allows you to assign geographic location data to a drawing file.

Shape Lite Pro Mechanical BIM

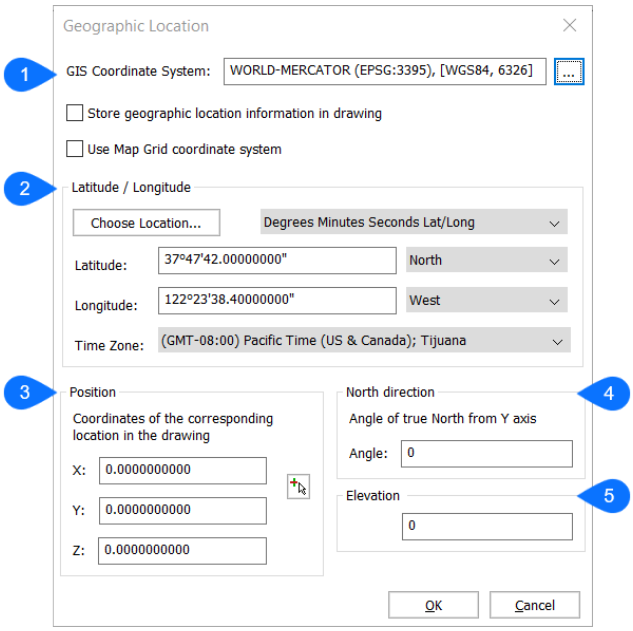
Icon:

Alias: GEO

12.16.1 Description

Opens the **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

12.16.2 GIS Coordinate System

Reports the GIS system being used by the drawing.

Note: Click the Browse button (...) to change the GIS Coordinate System.

Store geographic location information in drawing

When checked, the geographic marker (Survey location marker) is displayed to indicate the position of the Geographic Location point in the drawing.

Use Map Grid coordinate system

When enabled, the drawing is associated with real-world coordinates, the location data from GIS Coordinate System.

Note: When disabled, the origin point is determined by the values entered for X, Y, and Z, which correspond with the latitude and longitude.

12.16.3 Latitude / Longitude

Choose Location

Specifies the geographic location through 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 latitude. 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.

Note: This setting is saved to the TIMEZONE system variable.

12.16.4 Position

Specifies the position of the geographic location point in the drawing by entering coordinate values.

Note: You can also select a specific point in the workspace to set the position.

12.16.5 North direction

Specifies the angle of the sun from north, in the context of the world coordinate system (the angle between the True North and the Y-axis, measured clockwise).

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

12.16.6 Elevation

Specifies an elevation for the long-lat point. The value can be positive or negative.

12.16.7 About CSMAP

CSMAP is a coordinate system and map projection that allows users to access more geographic coordinates conversions and more precise geospatial analysis. Coordinate systems are a very important piece of mapping or geospatial coordinates, and are constantly updated.

You can download and install CSMAP coordinate system on demand:

1 Click here to download the CSMAP coordinate system.

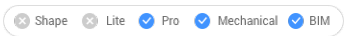
2 Unzip the archive and copy the content of the **coordinate_system** folder in the BricsCAD installation folder.

The default path is: *C:\Program Files\Bricsys\BricsCAD V25 en_US*.

3 The next time BricsCAD is started, the GEOGRAPHICLOCATION command will display a combined list of coordinate systems from **Geodatabase.xml** and **CoordinateSystemFiles** folder.

12.17 GEOIMPORT command

Imports a drawing with respect to the geographic location.



12.17.1 Description

Imports a drawing with respect to the geographic location in the source and the target drawing.

Note: The geographic location must be defined in both the source and the target drawing.

12.17.2 Method

This command displays the **Open drawing file** dialog box, which allows you to select either a Standard Drawing File (*.dwg) or a Drawing Exchange Format (*.dxf).

12.18 GEOMAP command

Sets the visibility of online maps.

Shape Lite Pro Mechanical BIM

12.18.1 Description

Displays a map from an online map service within the current drawing.

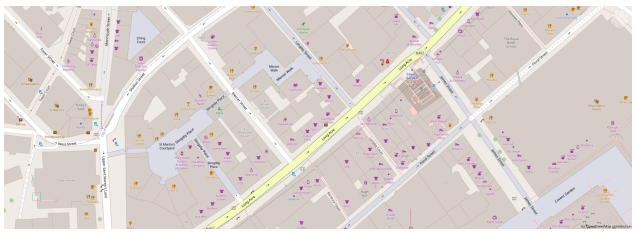
Note:

- Make sure the geographical location and coordinate system is set using the GEOGRAPHICLOCATION command.
- The map will only be available if an API key has been assigned for the appropriate online map service. For more details, refer to the **GEOMAPKEY command** article.

12.18.2 Options within the command

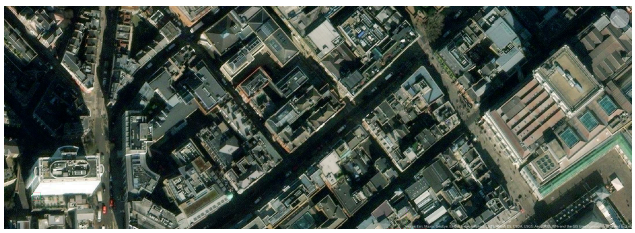
open street Map

The map is displayed in the **open street Map** map style.



esri Imagery

The map is displayed in the **Imagery** map style.



esri Streets

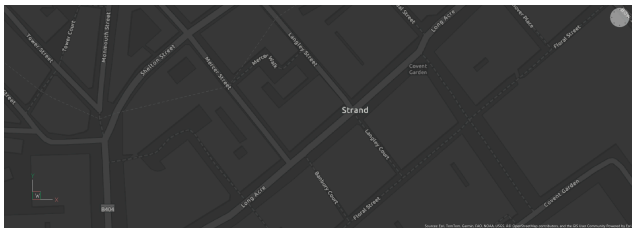
The map is displayed in the **Streets** map style.



esri Light
The map is displayed in the **Light** map style.



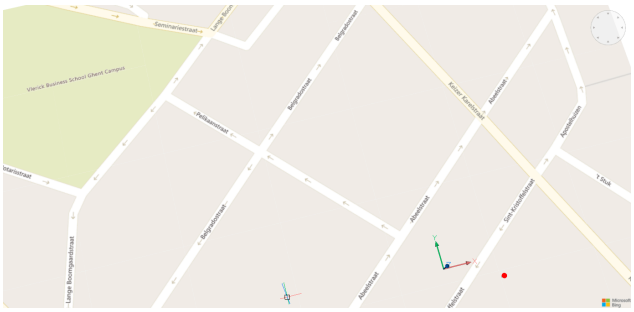
esri Dark
The map is displayed in the **Dark** map style.



bing Aerial
The map is displayed in the **Aerial** map style.



bing Road
The map is displayed in the **Road** map style.



bing Hybrid
The map is displayed in the **Hybrid** map style, combining **Aerial** with **Road** map styles.



Off
Hides the geographical map.

12.19 GEOMAPIMAGE command

Creates an image capture of online maps.

Shape Lite Pro Mechanical BIM

12.19.1 Description

Creates a rectangle image capture of the online map and embeds it in the drawing.

Note: You can use the GEOMAPIMAGETYPE, GEOMAPIMAGERESOLUTION and GEOMAPIMAGEUPDATE commands to further manage the map image.

12.19.2 Method

Select the area on the map of which you want to create a capture. An embedded image of the online maps is then created, and a frame is displayed on the map indicating the created image capture.

Note: Before creating a map image capture, you first have to run the GEOGRAPHICLOCATION command to define and store a geographic location for the current drawing and run the GEOMAP command to set an online maps style.



The size and/or the position of the map image frame can then be adjusted and the map image is updated accordingly.

Note: To keep only the map image capture visible, run the GEOMAP command again and select the **Off** option.



Note: The copyright and watermark are added in the bottom right corner of the map image.

12.19.3 Options within the command

Viewport
Sets the current viewport as frame for the capture image of online maps.

12.20 GEOMAPIMAGERESOLUTION command

Sets the resolution for a map image.

Shape Lite Pro Mechanical BIM

12.20.1 Description

Sets the resolution for the map image that was created with the GEOMAPIMAGE command.

12.20.2 Options within the command

Coarse
Sets the resolution of the map image to a less detailed view.

Optimal
Sets the resolution of the map image to an optimal detailed view.

Fine
Sets the resolution of the map image to a high detailed view.

VeryFine
Sets the resolution of the map image to a very high detailed view.

12.21 GEOMAPIMAGETYPE command

Sets the map style of a map image.

Shape Lite Pro Mechanical BIM

12.21.1 Description

Sets the map style of the map image that was created with the GEOMAPIMAGE command.

12.21.2 Options within the command

open street Map
The captured map image is displayed in the **open street Map** map style.

esri Imagery
The captured map image is displayed in the **Imagery** map style.

esri Streets
The captured map image is displayed in the **Streets** map style.

esri Light
The captured map image is displayed in the **Light** map style.

esri Dark
The captured map image is displayed in the **Dark** map style.

bing Aerial
The captured map image is displayed in the **Aerial** map style.

bing Road
The captured map image is displayed in the **Road** map style.

bing Hybrid
The captured map image is displayed in the **Hybrid** map style, combining **Aerial** with **Road** map styles.

Note: See the **GEOMAP command** article for more information about the map styles available in BricsCAD.

12.22 GEOMAPIMAGEUPDATE command

Updates a map image.

Shape Lite Pro Mechanical BIM

12.22.1 Description

Updates the map image that was created with the GEOMAPIMAGE command.

12.22.2 Options within the command

Optimize
Optimises the map image.

Reload
Reloads the map image.

12.23 GEOMAPKEY command

Allows you to enter API keys to access online map services.

Shape Lite Pro Mechanical BIM

12.23.1 Description

Opens the **Online Maps Support** dialog box, where you can enter API keys to access online map services.



Important:

- **Bing Maps Key**: Microsoft is phasing out Bing Maps and consolidating it under Azure Maps. New Bing Maps customers are no longer accepted. Bricsys will not support Azure Maps. Existing Bing Maps license holders can use Bing Maps in BricsCAD until June 30, 2025. Transitioning to Esri Maps is recommended for continued use.
- **Esri Maps Key**: An Esri Maps key provides access to Esri online maps and can be generated at ArcGIS Location Platform.

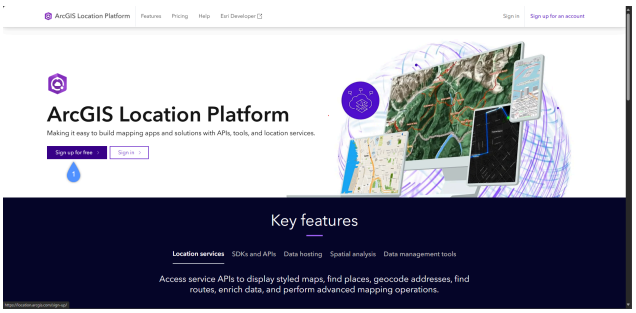
  For pricing information visit ArcGIS Location Platform pricing.

  Each month, the first 2 million tiles are free, after which a charge of $0.15 per 1,000 tiles applies. If the 2 million tile limit is exceeded, the maps will cease to function until the user either enables the pay-as-you-go option within their ESRI dashboard or purchases a voucher.

  Tip: To avoid exceeding the 2 million tile limit, it is advisable for each user to create their own ESRI account and API key.
- **OpenStreetMaps**: It does not require an API key and is free to use.

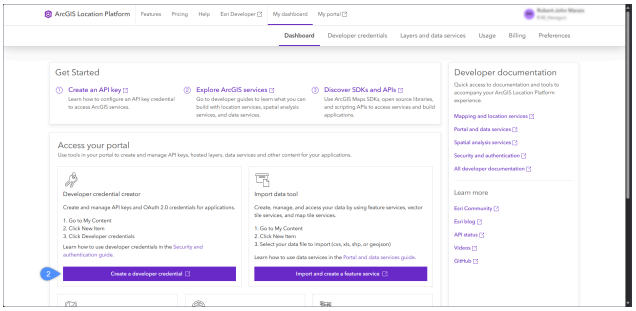
12.23.2 Configuring the Esri Maps Key

1 Sign up for a free account at ArcGIS Location Platform. Click the **Sign up for free** button (1), then fill in all required information.

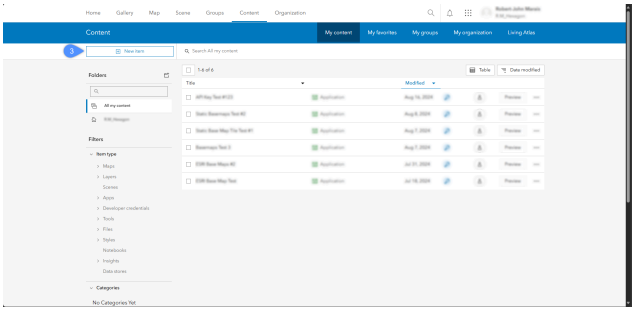


Note: Occasionally, Esri may block new sign-ups without a clear reason. If this happens, email accounts@esri.com to resolve the issue.

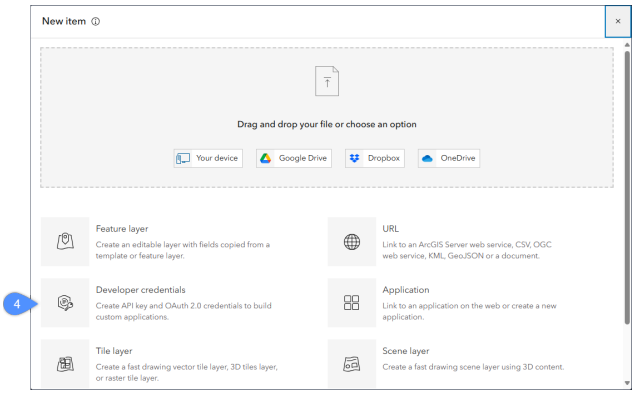
2 Once you have signed up, sign in, and on the ArcGIS Location Platform dashboard, select **Create a developer credential** (2), which opens a new window.



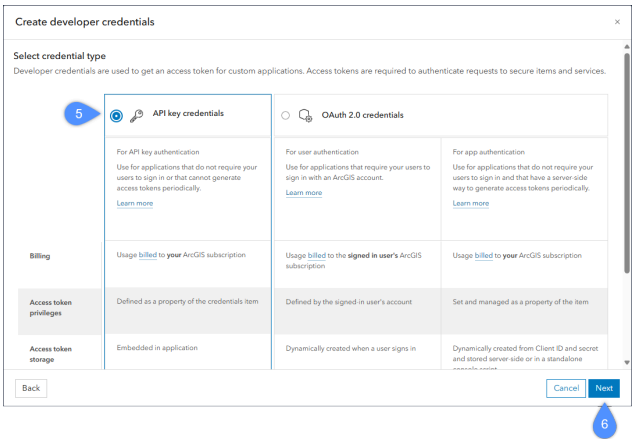
3 Choose **New item** (3).



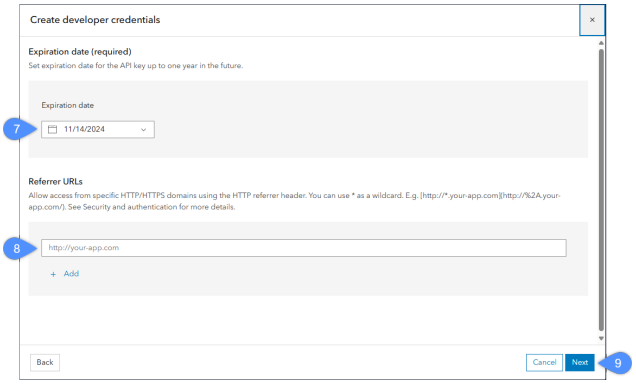
4 Select **Developer credentials** (4).



5 Choose **API key credentials** (5) and click **Next** (6).



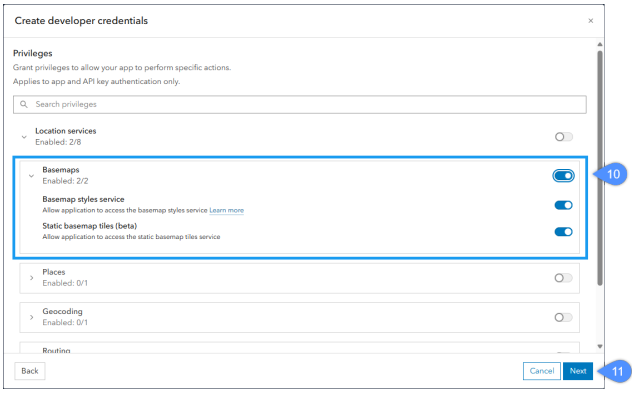
6 Set an **Expiration date** (7) for the API key. You can skip the **Referrer URLs** section (8). Click **Next** (9).



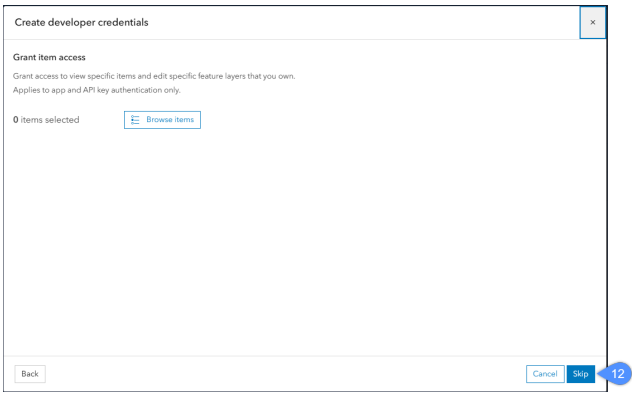
Note: A new API key will need to be generated once the current one expires.

7 Under **Privileges**, select **Basemaps** (10) and click **Next** (11).

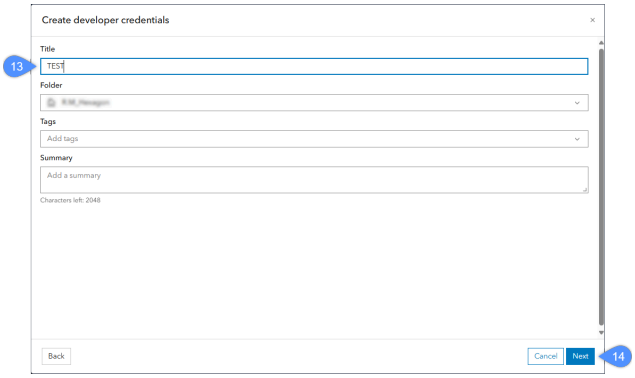
Note: Ensure that both **Basemap styles service** and **Static basemap tiles (beta)** are selected.



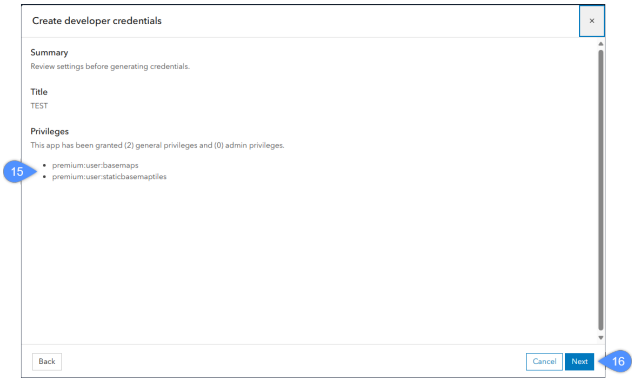
8 Click **Skip** (12) to skip the **Grant item access** step.



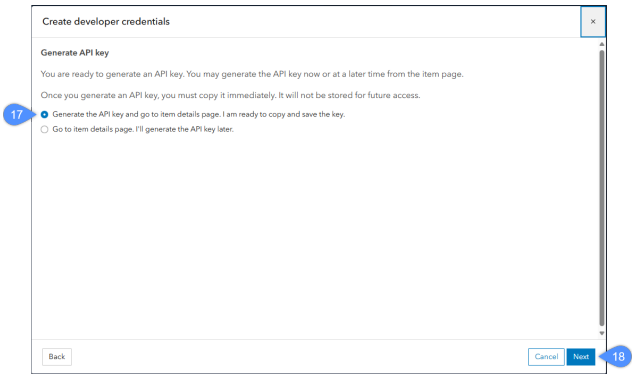
9 Enter a title (13). The rest of the fields can be skipped. Click **Next** (14).



10 In the Summary, ensure that the privileges include both **premium:user:basemaps** and **premium:user:staticbasemaptiles** (15). Click **Next** (16).

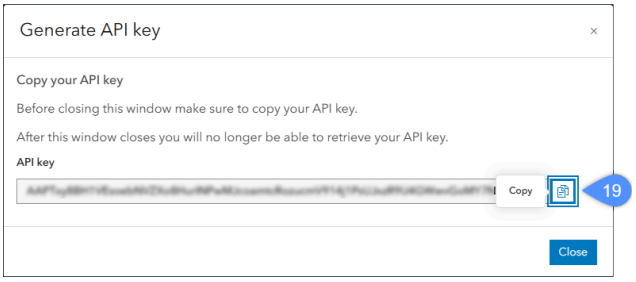


11 Select **Generate the API key and go to item details page. I am ready to copy and save the key** (17) and click **Next** (18).



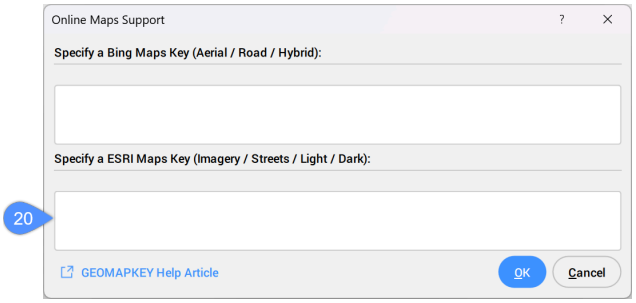
12 Copy the API key (19) and close the window.

13



Note: Be sure to copy the API key and save it in a secure location, as it will not be stored on your Esri account.

14 Enter the API key in the **Esri Key** field (20) within the **Online Maps Support** dialog box.



15 Run the GEOMAP command and select an Esri map.

16 The Command line prompts:

```
Select map type [open street Map/esri Imagery/esri Streets/esri Light/esri
 Dark/bing Aerial/bing Road/bing Hybrid/Off] <Off>:
```

12.24 GEOMAPMODE command

Reports the online maps style.

Shape Lite Pro Mechanical BIM

12.24.1 Description

Reports the online maps style used in the current viewport.

Note: You can change the map style with the GEOMAP command.

Reported value	0.: Online maps are not displayed. 1.: The online maps style is set to **bing Aerial**. 2.: The online maps style is set to **bing Road**. 3.: The online maps style is set to **bing Hybrid**. 4.: The online maps style is set to **esri Imagery**. 5.: The online maps style is set to **open street Map**. 6.: The online maps style is set to **esri Streets**. 7.: The online maps style is set to **esri Light**. 8.: The online maps style is set to **esri Dark**.

12.25 GEOMCONSTRAINT command

Applies geometric relationships between entities, on entities, and on valid constraint points.

Shape Lite Pro Mechanical BIM

12.25.1 Description

Constraints keep entities in a fixed position, such as perpendicularly or vertically.

Note: Geometric constraints can be applied to the following entities and their constraint points:

Entity Type	Valid Constraint Points
Lines	Endpoints, midpoint
Arcs, elliptical arcs	Endpoints, center point, midpoint
Circles, ellipses	Center point
Polyline segments	Endpoints, vertices, midpoints
Polyline arcs	Endpoints, vertices, midpoints, center points
Splines	Endpoints
Inserted entities: blocks, xrefs, text, mtext, attributes, tables	Insertion points

12.25.2 Options within the command

Horizontal

Constrains entities or pairs of points to lie parallel to the X-axis of the current coordinate system. See the GCHORIZONTAL command.

Vertical
Constrains entities or pairs of points to lie parallel to the Y-axis of the current coordinate system. See the GCVERTICAL command.

Perpendicular
Constrains two entities to lie perpendicularly to each other. See the GCPERPENDICULAR command.

PArallel
Forces two entities to be parallel to one other. See the GCPARALLEL command.

Tangent
Constrains two entities to maintain a point of tangency to each other or their extensions. See the GCTANGENT command.

SMooth
Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. See the GCSMOOTH command.

Coincident
Applies a coincident geometrical constraint to two points or constrains a point to an entity. See the GCCOINCIDENT command.

CONcentric
Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. See the GCCONCENTRIC command.

COLlinear
Forces entities to be collinear. See the GCCOLLINEAR command.

Symmetric
Constrains two entities or points to lie symmetrically with respect to a selected line. See the GCSYMMETRIC command.

Equal
Constrains circular entities to the same radius, or linear entities to the same length. See the GCEQUAL command.

Fix
Constrains points and entities at a fixed position. See the GCFIX command.

12.26 GISCONVERT command

Converts Object Data in the current drawing from Autodesk® Civil 3D to BricsCAD GIS data.

Shape Lite Pro Mechanical BIM

12.26.1 Description

Converts Object Data (lines, polylines, points, or blocks) and Mpolygons in the current drawing from Autodesk® Map 3D or Civil 3D to BricsCAD GIS data.

The BricsCAD GIS data is displayed in the **Properties** panel and in the **GIS Attribute table** dialog box.

A GIS Layer is created in the GIS tab from the **Civil Explorer** panel.

12.27 GISEXPORT command

Exports vector geographic features to a SHP or KML/KMZ file formats.

Shape Lite Pro Mechanical BIM

Icon: GIS

12.27.1 Method

Select the entities you want to export and press **Enter**. The **Export file** dialog box opens to export vector geographic features with their original location, shape and attributes to a SHP or KML/KMZ file.

12.28 GISIMPORT command

Imports vector geographic features from SHP, KML/KMZ, or Geodatabase file formats.

Shape Lite Pro Mechanical BIM

Icon: GIS

12.28.1 Description

Allows you to import vector geographic features with their original location, shape and attributes from SHP, KML/KMZ, or Geodatabase files into the current drawing and specify properties for each imported GIS Feature (Layer).

12.28.2 Method

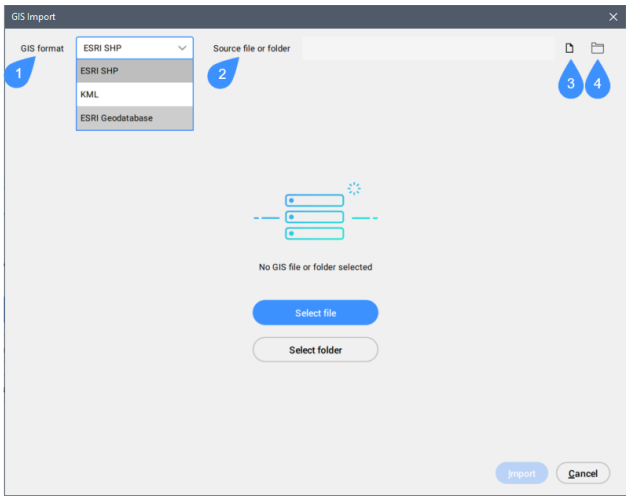
Select the GIS files in the **GIS Import** dialog box.

The files to import are displayed in the **GIS import** dialog box, where you can modify their properties, such as drawing layer, GIS Data, elevation and import area.

Note:

- GIS Features are imported in the drawing as CAD entities: points, lines, polylines.
- GIS Features are imported on specified drawing layers.
- Only GIS Features inside the specified area are imported in a drawing.
- ESRI Geodatabase formats are supported.
- The coordinate system transformation is enabled.

The **GIS Import** dialog box allows you to import different GIS Features (Layers) with shapes and attributes from SHP, KML/KMZ, or Geodatabase files into the current drawing. You can also specify the properties for each imported Feature.



1 GIS format

2 Source file or folder

3 Select the source file

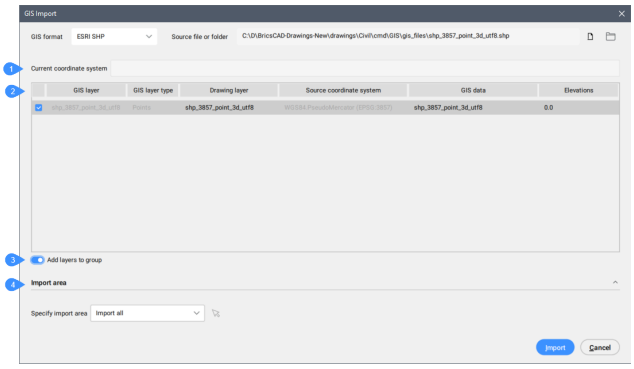
4 Select the source folder

GIS format
Defines the GIS format.

Source file or folder
Displays the file source path.

Select the source file
Opens the **Open file** dialog box to select a GIS file.

Select the source folder
Opens the **Choose input folder** dialog box to select a folder that contains the GIS files.



1 Current Coordinate System

2 Table of imported GIS layers

3 Add layers to group

4 Import Area

5 Import

12.28.3 Current Coordinate System

Reports the GIS coordinate system being used by the current drawing.

12.28.4 Table of imported GIS Features

Lists the GIS Layers to import and allows you to define their properties.

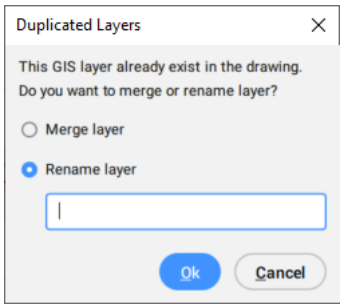
Selected
Allows you to select/deselect GIS features to import.

Note: A warning message appears when nothing is selected for import.

GIS layer
Specifies the name of the GIS layer.

Note: Importing a duplicate GIS layer by renaming or merging it is possible. Follow the indications from the warning message.

Press the ⚠ button to open the Duplicated Layers dialog box.



GIS layer type
Specifies the layer type, for example lines, polygons, points, etc.

Drawing layer
Specifies the drawing layer on which the entities are created.
Click the **Drawing layer** field of the GIS feature to open the **Drawing Layer** dialog box to set the layer.

Note: Importing a duplicate GIS layer by renaming or merging it is possible. Follow the indications from the warning message.

Existing layer
Allows you to choose an existing layer from current drawing.

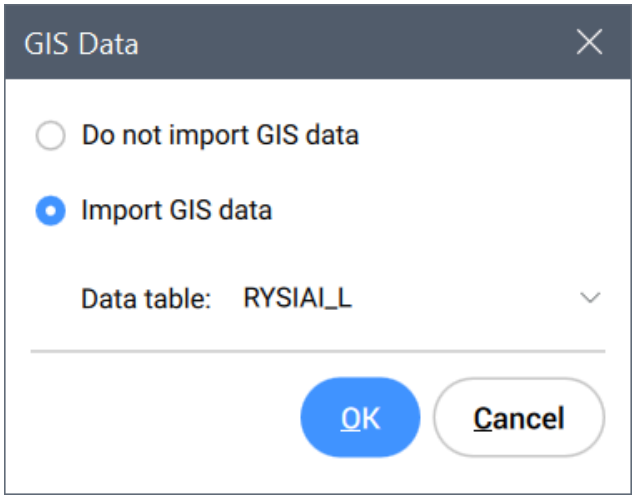
New layer
Allows you to create a new layer.

Layer from data field
Allows you to choose a layer from the data field.

Source Coordinate System
Specifies the source coordinate system that is used in the imported GIS file, which is read-only.

GIS Data
Specifies the GIS Data.

Click the **GIS Data** field of the GIS feature to open the **GIS Data** dialog box where you can choose whether or not to import the GIS data.



Do not import GIS data
If selected, the GIS data is not imported.

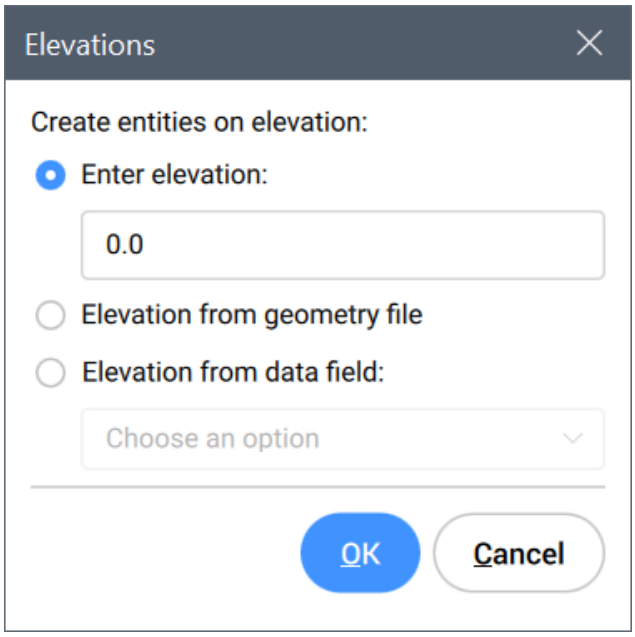
Import GIS data
If selected, it imports the GIS data from the specified data table.

Note: When an entity with GIS data is copied, the GIS data is transferred with it.

GIS data linked to entities is maintained when:

- Entity with GIS data is copied in the current drawing.
- You create a block definition as an external DWG file using the WBLOCK command.
- Entity with GIS data is moved between drawings using the COPY and PASTE commands.

Elevations
Specifies the elevation at which entities are created.
Click the **Elevations** field of the GIS feature to open the **Elevations** dialog box to set the elevation.



Enter elevation
Allows you to enter the elevation value.

Elevation from geometry file
Uses the elevation from the geometry file.

Elevation from data field
Uses the elevation from the specified data field.

12.28.5 Add layers to group

Adds the imported GIS layers to a new group, visible under the **GIS** tab from the **Civil Explorer** panel.

12.28.6 Import Area

Allows you to specify the import area.

Specify import area

Import All
Uses the entire drawing as import area.

Select clipping polygon

Click the mouse icon to select a polygon in the drawing that defines the import area.

Draw clipping polygon

Click the mouse icon to specify an import area in the drawing.

12.28.7 Import

Imports the GIS features in the drawing and closes the **GIS Import** dialog box.

12.29 -GISIMPORT command

Imports vector geographic features from SHP or KML/KMZ file formats via Command line.

Shape Lite Pro Mechanical BIM

12.29.1 Description

Allows you to import vector geographic features with their original location, shape and attributes from SHP or KML/KMZ files into the current drawing and specify properties for each imported GIS Feature (Layer).

Note:

- GIS Features are imported in the drawing as CAD entities: points, lines, polylines.
- GIS Features are imported on specified drawing layers.
- Only GIS Features inside the specified area are imported in a drawing.
- ESRI Geodatabase formats are supported.

12.29.2 Method

Select the import file or folder. Then, insert the full path for the file or folder you chose.

12.29.3 Options within the command

File
Opens the **Open file** dialog box to select a file to import.

fOlder
Opens the **Choose input folder** dialog box to select a folder to import.

All
Allows you to select all the entities from the file or folder to import.

Select
Allows you to select any closed entity made from polylines that is non-self-intersecting to define the import area.

Define
Allows you to define any closed entity made from polylines that is non-self-intersecting to define the import area.

Undo
Undoes the last polygonal segment.

Cancel
Exits the command.

12.30 GETSEL command (Express Tools)

Creates a selection set of entities based on entity type and layer filters.

Shape Lite Pro Mechanical BIM

Icon: 

12.30.1 Method

Select an entity on the source layer, then select an entity of the type you want.

For example, if you select at the first prompt an entity on NewLayer1 and at the second prompt a circle, the GETSEL command collects all circle entities on layer NewLayer1.

Note:

- The entities are placed in the current selection set.
- These entities can be selected using the SELECT command and entering **P**.

12.31 GOTOSTART command

Opens the **Start** page.

Shape Lite Pro Mechanical BIM

12.31.1 Description

Opens or activates the **Start** page.

In the **Start** page you can:

- Create new drawings based on a specified template or open recent drawings through the **Start** tab.

  Note: The number of available recent files is controlled by the value of the RECENTFILES system variable.

- Access BricsCAD tutorials and sample drawings through the **Learn** tab.

  Note: Internet access is required.

- Access the Third Party Application catalog on the Bricsys website through the **Apps** tab.

  Note: Internet access is required.

- Have an overview of your most used commands and achievements through the **Insights** tab.

  Note: You need to join the BricsCAD Analytics Program to get insights.

12.32 GRADIENT command

Opens the **Hatch and Gradient** dialog box.

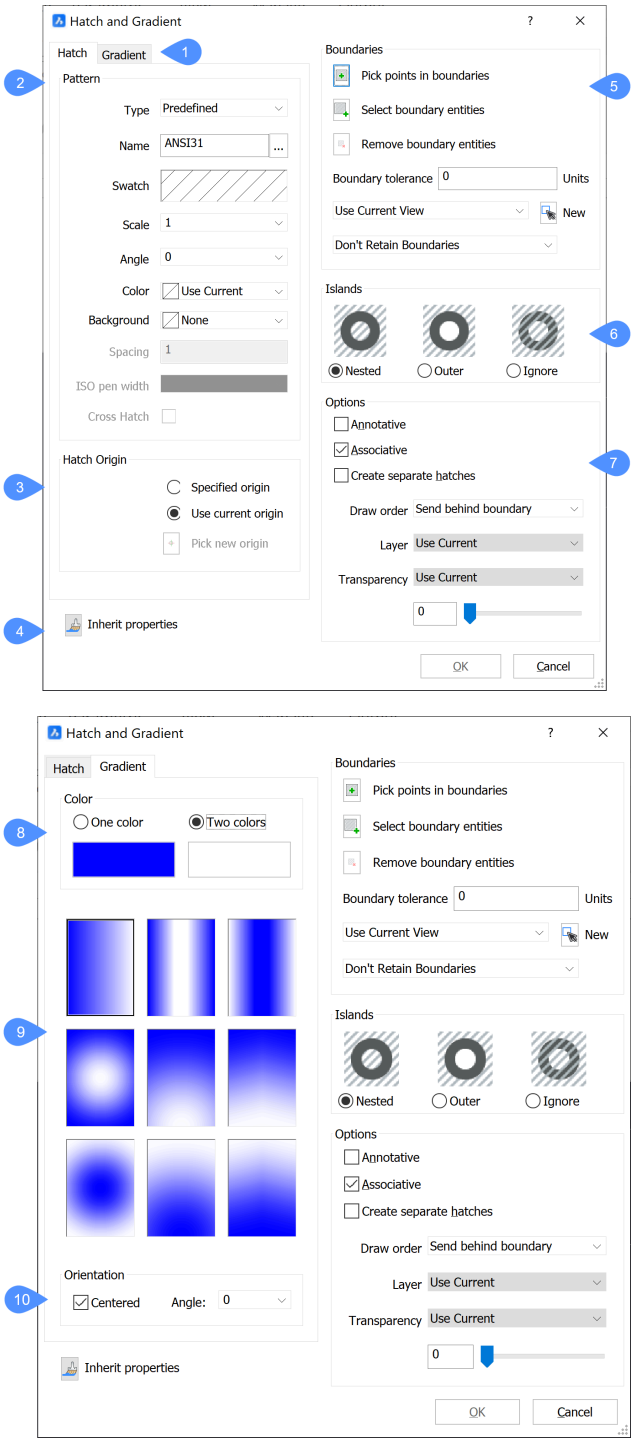
Shape Lite Pro Mechanical BIM

Icon: 

12.32.1 Description

Opens the **Hatch and Gradient** dialog box to create a hatch entity in the current drawing.

The **Hatch and Gradient** dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.



1 Hatch and Gradient tabs

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

12.32.2 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.

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

Swatch
Displays the **Hatch Pattern Palette** dialog box.

Note: 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.

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

Angle
Sets the angle of the pattern.

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

Color
Sets the color of the hatch pattern lines.

Note: For the **Use Current** 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.

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

ISO pen width
Specifies the lineweight.

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

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

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

12.32.3 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

12.32.4 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

12.32.5 Boundaries

Specifies the boundary conditions of the hatch.

Pick points in boundaries
Specifies the closed areas (boundaries) in which to place the pattern.

Note: 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.

Note: 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.

Note: 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.

Note: 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 Superior versions only).

12.32.6 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.

12.32.7 Options

Annotative
Toggles the Annotative property of the hatch. When turned On, BricsCAD applies the current annotative scale.

Note: 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.

Note: 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.

Note: For the **Use current** option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

12.32.8 Gradient Color

Specifies the gradient color. The gradient can include either one color or two colors to create a solid fill hatch.

12.32.9 Gradient pattern

Specifies the pattern of the gradient.

12.32.10 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.

12.33 -GRADIENT command

Fills closed areas with a gradient fill.

Shape Lite Pro Mechanical BIM

12.33.1 Description

Fills closed areas with solid fills in gradients of one or two colors in a variety of patterns.

Note: 3D entities cannot be filled in gradients.

12.33.2 Method

There are two methods to fill in gradients:

- Single color
- Two colors

12.33.3 Options within the command

hatch Properties
Allows you to specify a hatch pattern.

? to list patterns
Lists the names of all hatch patterns. Press F2 to see the list in the prompt history window.

Solid fill
Specifies a solid-color filled hatch.

User defined
Allows you to customize the hatch pattern.

Set Angle for lines
Specifies the angle for the pattern lines.

Space between standard pattern lines
Specifies the space between the hatch lines.

Cross-hatch area?
Toggles between **Yes** and **No**. If Yes, the pattern is repeated at 90 degrees to the original.

Gradient
Allows you to customize a gradient pattern.

Enter a gradient name
Allows you to specify a gradient pattern name.

Enter an angle for the gradient
Allows to specify an angle for the gradient fill.

Center the gradient?
Toggles between **Yes** or **No**.

? to list patterns
Lists the names of all gradient patterns. Press F2 to see the list in the prompt history window.

Single color
Specifies a new gradient color and tint.

Enter a shade or tint value
Specifies a number between 0 and 1 (0 = dark, 1 = light).

Two color
Specifies two new gradient colors.

Truecolor
Allows you to specify the true color for the layers in selected viewports, by entering the values for Red, Green, and Blue.

COlorbook
Opens a color book by entering its name and allows you to specify a color name from the loaded color book.

Note: The COLORBOOKPATH system variable specifies the folder(s) in which BricsCAD should look for color book files.

Select entities
Allows you to selects the entities that make up the hatching boundary to constrain the extent of the pattern.

Remove boundaries
Removes the boundary for the selected entities.

Advanced options
Allows you to set advanced options for the hatch.

Boundary set
Specifies the entities to consider while creating boundaries.

Retain boundary
Determines whether the temporary boundary is retained after the command ends.

Island detection
Toggles whether islands are hatched over or not. Islands represent interior boundaries.

Specify style
Specifies how islands are treated.

Nested
Alternating islands are hatched, beginning with the outermost one.

Outer
Only the outermost region is hatched. Interior islands are not hatched.

Ignore
Islands are ignored and hatched over.

set Associativity
Toggles whether hatches are associative, so they do update/not update their geometry alongside the boundaries.

boundary Tolerance
Specifies the largest gap the program ignores when hatching a boundary that is not fully closed.

Note: A value of 0 means that no gaps in the boundary are tolerated by the program.

separate Hatches
Allows you to create separate hatches for each closed area or a single hatch entity for all.

Draw order
Specifies whether the hatch pattern appears visually above/below overlapping entities or its boundary.

Origin
Specifies a new value for the hatch origin.

ANnotative
Sets the annotative property for the hatch. It applies the current annotative scale, defined by the CANNOSCALE system variable.

Layer
Specifies the layer on which the hatch should be placed.

Transparency
Specifies a value between 0 and 90 for the transparency.

Note: A value of 0 means fully opaque. The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off.

ByLayer
Applies the value of the transparency property of the layer on which the hatch resides.

ByBlock
The transparency value is controlled by block.

Use current
Applies the current transparency value as defined by the CETRANSPARENCY system variable.

Note: The transparency value for new hatches is saved by the HPTRANSPARENCY system variable.

Undo
Removes selected boundaries from the selection.

12.34 GRADIENTBKGOFF command

Turns off the background gradient.

Shape Lite Pro Mechanical BIM

12.34.1 Description

Turns off the background gradient colors (used by all visual styles, except for 2D Wireframe) to display the default background color.

12.35 GRADIENTBKGON command

Turns on the background gradient.

Shape Lite Pro Mechanical BIM

12.35.1 Description

Turns on the background gradient colors, which are used by all visual styles (except for 2D Wireframe).

12.36 GRADING command

Creates a graded surface on existing TIN surfaces.

Shape Lite Pro Mechanical BIM

Icon: 

12.36.1 Description

Creates a graded surface between a selected entity and a TIN surface or an offset or slope graded surface from a selected entity. Select the entity to define a graded surface, select the target TIN surface to create a graded surface and move the mouse to adjust the slope of the grading.

12.36.2 Method

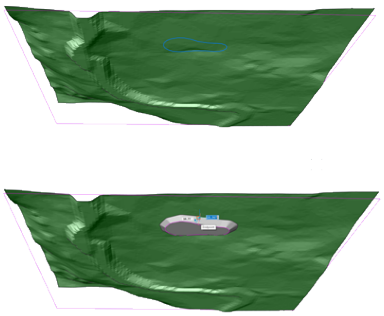
Grading can be created along the entire length of the selected input entity or between specified start and end point.

Note: Gradings also support transitions.

12.36.3 Options within the command

slOpe/offset
Sets either the slope or offset option of the grading.

Note: The offset-slope projection method requires input entity, offset, and slope.



Entire length
Creates a grading using the entire input entity.

Note: Sharp corners are also supported. Change the **Corner type** property to **Sharp** in the **Properties** panel.

12.37 GRADINGBALANCE command

Balances grading cut and fill volumes.

Shape Lite Pro Mechanical BIM

Icon:

12.37.1 Description

Balances grading 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.

12.38 GRADINGEDIT command

Splits or merges gradings.

Shape Lite Pro Mechanical BIM

Icon:

12.38.1 Description

Merges two gradings into one with several regions with (automatic) calculation of transitions between gradings, or splits the grading into multiple regions so that the slopes for each region can be edited separately.

Select a grading to be split into multiple regions, or to be merged with another grading.

12.38.2 Options within the command

Split
Splits the grading into multiple regions, so the slopes for each region can be edited separately by moving the grips.

Merge
Merges two gradings into one.

Switch
Switches the direction.

12.39 GRAPHICOVERRIDE command

Temporarily modifies the appearance of entities in 3D model space.

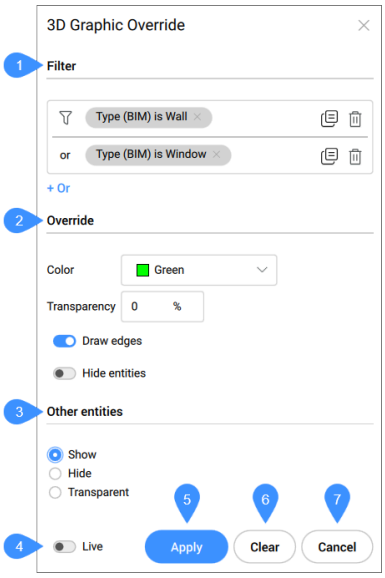
Shape Lite Pro Mechanical BIM

Icon:

12.39.1 Method

GRAPHICOVERRIDE is a 3D drawing customization feature that allows you to temporarily override the visual appearance of a specific selection of objects based on BIM properties.

The command opens the **3D Graphic Override** command panel to allow you to define which entities in the 3D model space will be overridden and how their appearance should be modified.



1 Filter

2 Override

3 Other entities

4 Live

5 Apply

6 Clear

7 Cancel

Filter
Allows you to define which entities in the 3D model space will be overridden by setting one or multiple criteria based on BIM properties, quantities, or custom properties. Filter rows can be added to filter by multiple parameter combinations. These rows are separated using the logical "OR" operation. The rows can be copied by clicking the copy symbol or deleted by clicking the delete symbol.

Override
Allows you to define how the filtered entity will be visualized.

Color
Allows you to select the color from the drop-down list with presets, or from the more extensive **Color** dialog box.

Transparency
Sets a transparency level for the selected entities.

Draw edges
Tick the option to draw the edges for the filtered selection.

Hide entities
Tick the option to hide the filtered selection.

Other entities
Allows you to define how all other entities, that do not meet the filter criteria, will be shown in the drawing.

Show
Shows all the other entities in the drawing as normal.

Hide
Hides all other entities that do not meet the filter criteria, which will visually isolate your selection.

Transparent
All other entities will become transparent. This will also put a clear focus on your selection, but you will still be able to see the other building elements for a visual context.

Live
By enabling the **Live** mode, every change you make to the settings is immediately applied to the model.

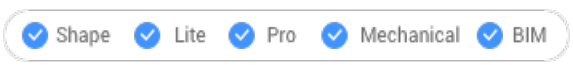
Apply
Applies the current settings to the 3D model of the opened document. This will overwrite any existing override on that model.

Clear
Clears any existing override of the current open document.

Cancel
Closes the command panel keeping the override active on the 3D model.

12.40 GRAPHSCR command

Switches to the graphic screen.



12.40.1 Description

Switches from the programs Prompt History window to the drawing window.

Prompt History window is displayed by the TEXTSCR command or by the F2 key.

12.41 GRID command

Toggles the display of the grid and sets some of its properties.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: G

12.41.1 Method

Specify the grid spacing value. It makes the x and y grid spacing the same.

12.41.2 Options within the command

ON
Turns on the display of the grid.

OFf
Turns off the display of the grid.

Snap
Synchronizes the grid spacing with the current snap spacing, so that the two are the same.

Aspect
Sets the aspect ratio of the grid by specifying the horizontal and vertical spacing.

12.42 GROUP command

Opens the **Entity grouping** dialog box.

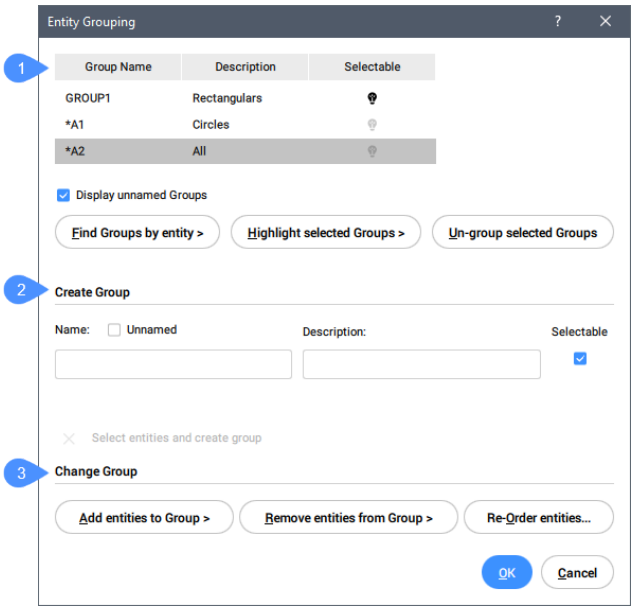
Shape Lite Pro Mechanical BIM

Icon: 

12.42.1 Description

Opens the **Entity grouping** dialog box.

The **Entity Grouping** dialog box allows you to view, create, modify and delete named groups of entities in the current drawing.



1 Existing groups

2 Create Group

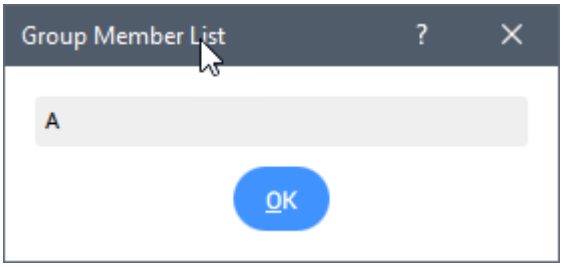
3 Change Group

12.42.2 Existing groups

Lists all existing groups with the Group Name and a short description.

Display unnamed Groups
Toggles the displaying of unnamed groups.

Find Groups by entity
Reports the name(s) of groups in the **Group Member List** dialog box when entities in the drawing are selected.



Highlight selected Groups
Highlights all entities that belong to the selected group.

Un-group selected Groups
Removes the group status from selected entities.

12.42.3 Create Group

Name
Specifies the name of a group.

Unnamed
Toggles whether the group has a specific 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.

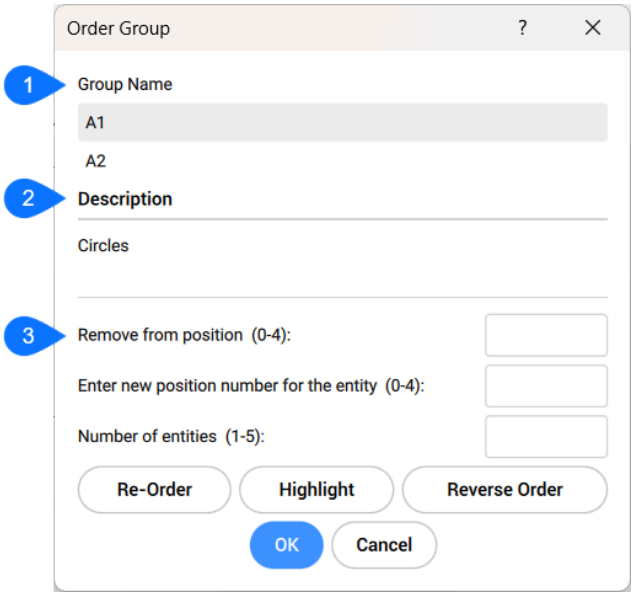
Select entities and create group
Selects entities in the workspace which should be part of the group by pressing the right side cross button.

12.42.4 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 through the **Order Group** dialog box.



1 Group Name

2 Description

3 Options

Group Name
Lists the names of named and unnamed groups.

Description
Displays the description of the selected group.

Options
Specifies several options

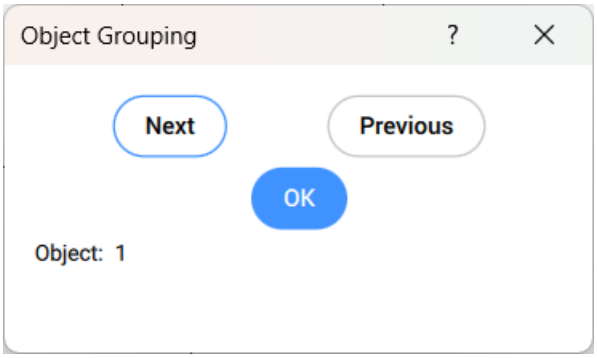
Remove from position
Specifies the position number of the entity to be reordered.

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 one by one by pressing the **Next** and **Previous** button from the **Object Grouping** dialog box.
Pressing **OK** button returns to the **Order Group** dialog box.



Reverse Order
Reverses the order of entities in the group.

12.43 -GROUP command

Creates and modifies groups of entities.

Shape Lite Pro Mechanical BIM

12.43.1 Description

Creates and modifies named groups of entities at the Command line.

12.43.2 Options within the command

?
Lists named and unnamed groups in the drawing.

Note: Unnamed groups are prefixed with *A, followed by an incremented number, such as *A3.

Order
Reverses the order of entities in a group.

Add
Adds entities to a group.

Remove
Removes entities from a group.

Explode
Ungroups the selected group.

reName
Renames groups.

Selectable
Toggles the selectivity of groups.

Create
Creates new groups.

13. H

13.1 HATCH command

Opens the **Hatch and gradient** dialog box.

Shape Lite Pro Mechanical BIM

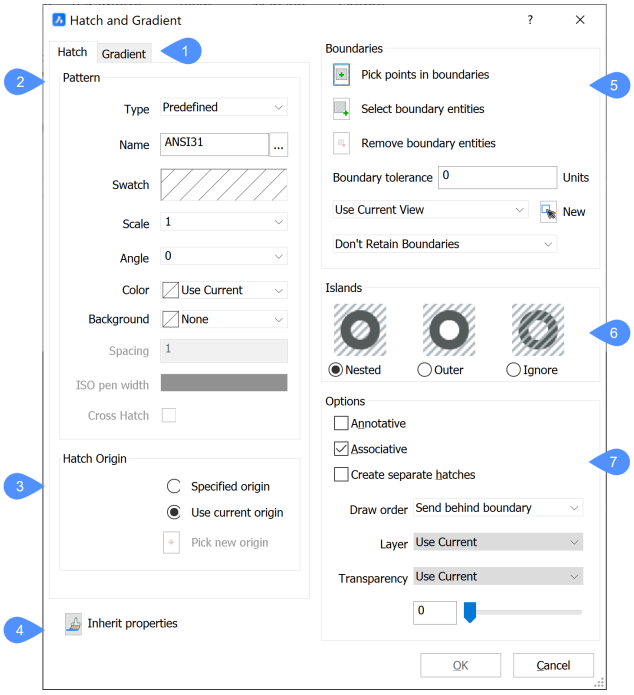
Icon:

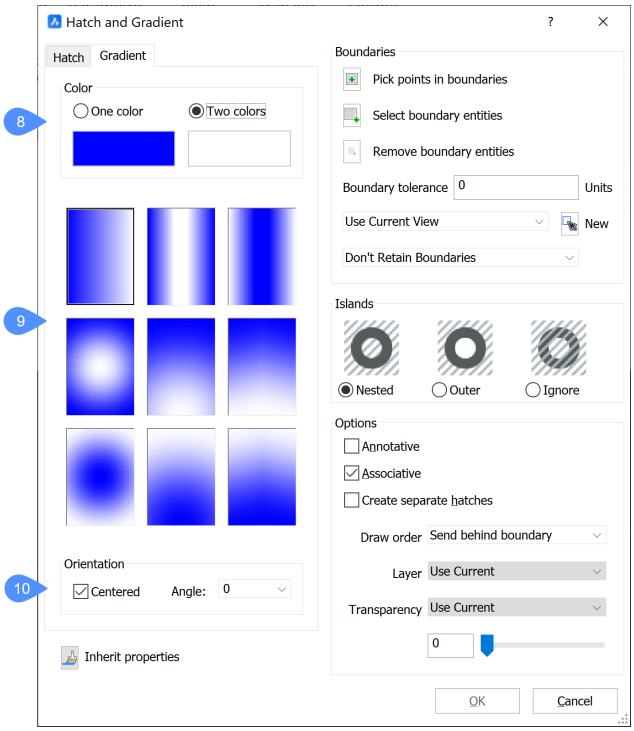
Alias: BH, H

13.1.1 Description

Opens the **Hatch and gradient** dialog box to create a hatch entity in the current drawing.

The **Hatch and Gradient** dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.





1 Hatch and Gradient tabs

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

13.1.2 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.

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

Swatch
Displays the **Hatch Pattern Palette** dialog box.

Note: 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.

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

Angle
Sets the angle of the pattern.

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

Color
Sets the color of the hatch pattern lines.

Note: For the **Use Current** 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.

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

ISO pen width
Specifies the lineweight.

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

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

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

13.1.3 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

13.1.4 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

13.1.5 Boundaries

Specifies the boundary conditions of the hatch.

Pick points in boundaries
Specifies the closed areas (boundaries) in which to place the pattern.

Note: 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.

Note: 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.

Note: 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.

Note: 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 Superior versions only).

13.1.6 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.

13.1.7 Options

Annotative
Toggles the Annotative property of the hatch. When turned On, BricsCAD applies the current annotative scale.

Note: 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.

Note: 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.

Note: For the **Use current** option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

13.1.8 Gradient Color

Specifies the gradient color. The gradient can include either one color or two colors to create a solid fill hatch.

13.1.9 Gradient pattern

Specifies the pattern of the gradient.

13.1.10 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.

13.2 -HATCH command

Fills closed 2D areas with repeating patterns or solid colors.

Shape Lite Pro Mechanical BIM

Alias: -BH, -B

Note: 3D entities cannot be hatched.

13.2.1 Method

There are two methods to apply a hatch pattern:

- Specify internal point.
- Select entities.

Note: Entities which are not visible in the current viewport when the command is launched cannot be selected. Partly visible entities can, however, be selected.

13.2.2 Options within the command

hatch Properties
Allows to specify a hatch pattern.

? to list patterns
Lists the names of all hatch patterns. Press F2 to see the list in the prompt history window.

Solid fill
Specifies a solid-color filled hatch.

User defined
Allows you to customize the hatch pattern.

Set Angle for lines
Specify the angle for the pattern lines.

Space between standard pattern lines
Specify the space between the hatch lines.

Cross-hatch area?
Toggles between **Yes** and **No**. If Yes, the pattern is repeated at 90 degrees to the original.

Gradient
Allows to specify a gradient pattern.

Enter a gradient name
Allows you to specify a gradient pattern name.

Enter an angle for the gradient
Allows to specify an angle for the gradient fill.

Center the gradient?
Toggles between **Yes** or **No**.

? to list patterns
Lists the names of all gradient patterns. Press F2 to see the list in the prompt history window.

Single color
Specifies a new gradient color and tint.

Enter a shade or tint value
Specifies a number between 0 and 1 (0 = dark, 1 = light).

Two color
Specifies two new gradient colors.

Truecolor
Allows you to specify the true color for the layers in selected viewports, by entering the values for Red, Green, and Blue.

COlorbook
Opens a color book by entering its name and allows you to specify a color name from the loaded color book.

Note: The COLORBOOKPATH system variable specifies the folder(s) in which BricsCAD should look for color book files.

Select entities
Allows you to selects the entities that make up the hatching boundary to constrain the extent of the pattern.

Remove boundaries
Removes the boundary for the selected entities.

Advanced options
Allows you to set advanced options for the hatch.

Boundary set
Specifies the entities to consider while creating boundaries.

Retain boundary
Determines whether the temporary boundary is retained after the command ends.

Island detection
Toggles whether islands are hatched over or not. Islands represent interior boundaries.

Specify style
Specifies how islands are treated.

Nested
Alternating islands are hatched, beginning with the outermost one.

Outer
Only the outermost region is hatched. Interior islands are not hatched.

Ignore
Islands are ignored and hatched over.

set Associativity
Toggles whether hatches are associative, so they do update/not update their geometry alongside the boundaries.

boundary Tolerance
Specifies the largest gap the program ignores when hatching a boundary that is not fully closed.

Note: A value of 0 means that no gaps in the boundary are tolerated by the program.

separate Hatches
Specifies to create separate hatches for each closed area or a single hatch entity for all.

Draw order
Specifies whether the hatch pattern appears visually above/below overlapping entities or its boundary.

Origin
Specifies a new value for the hatch origin.

ANnotative
Sets the annotative property for the hatch. It applies the current annotative scale, defined by CANNOSCALE system variable.

Layer
Specifies the layer on which the hatch should be placed.

Transparency
Specifies a value between 0 and 90 for the transparency.

Note: A value of 0 means fully opaque. The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off.

ByLayer
Applies the value of the transparency property of the layer on which the hatch resides.

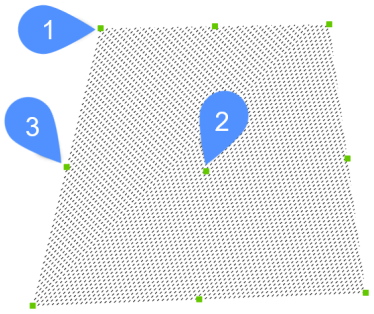
ByBlock
The transparency value is controlled by block.

Use current
Applies the current transparency value as defined by the CETRANSPARENCY system variable.

Note: The transparency value for new hatches is saved by the HPTRANSPARENCY system variable.

Undo
Removes selected boundaries from the selection.

Note: Hatches can be edited directly through grips.



- Drag the central grip (2) to move the hatch.
- Drag the vertex (1) or a midpoint grip (3) to change the hatch boundary.

13.3 HATCHEDIT command

Edits hatches through a dialog box.

Shape Lite Pro Mechanical BIM

Icon:

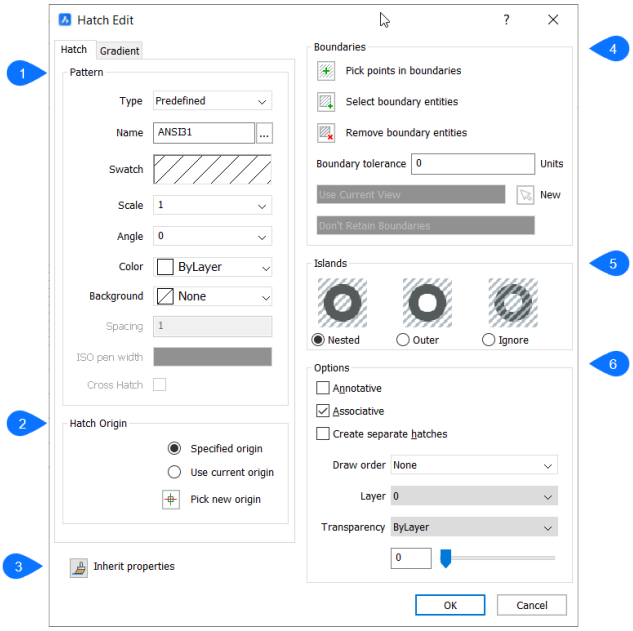
Alias: HE

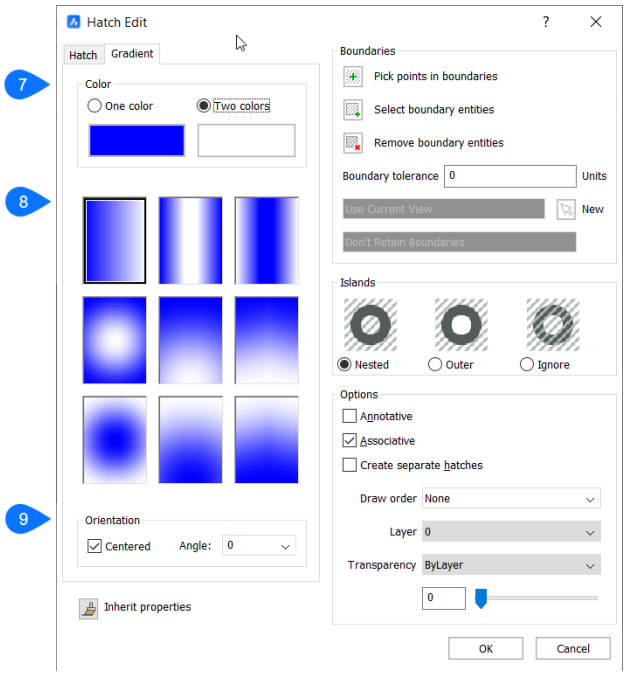
13.3.1 Description

Selects a hatch pattern or a gradient fill and edit them through **Hatch Edit** dialog box.

Note: The **Properties** panel can also be used to edit hatches and gradient fills.

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

13.4 -HATCHEDIT command

Edits the boundaries of hatch entities at the Command line.

Shape Lite Pro Mechanical BIM

13.4.1 Description

Edits the boundaries of hatch and gradient entities at the Command line.

13.4.2 Method

There are two methods to use -HATCHEDIT command:

- Select a hatch entity.
- Select a gradient entity.

13.4.3 Options within the command

Disassociate
Removes the associative property of the selected hatch or gradient entity.

ADd boundaries
The area of a hatch is modified by adding boundaries.

Specify internal point
The boundary is determined from an existing enclosed area around the specified point. The selected hatch is then associated with that boundary.

Select entities
The boundary is determined from the selected entities that form an enclosed area. The selected hatch is then associated with that boundary.

Remove boundary
The area of a hatch is modified by removing boundaries.

ASsociate
Associate the selected hatch or gradient entity to a different boundary set.

13.5 HATCHEDITEXT command

Edits the vertices of a hatch or gradient entity.

Shape Lite Pro Mechanical BIM

13.5.1 Description

Edits the vertices of a hatch or gradient entity interactively through the command line.

13.5.2 Options within the command

Add vertex
Adds vertices by stretching selected grip points.

Remove vertex
Removes vertices.

convert to Line
Converts arc segments to line segments.

convert to ARc
Converts line segments to arc segments.

13.6 HATCHGENERATEBOUNDARY command

Generates a boundary around a hatch or gradient fill.

Shape Lite Pro Mechanical BIM

Icon: 

13.6.1 Description

Creates a boundary made of a polyline around the selected hatches or gradient fill.

13.7 HATCHTOBACK command

Moves all hatch entities in the drawing behind all other overlapping entities.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command selects all hatch patterns in the current drawing automatically.

13.8 HELIX command

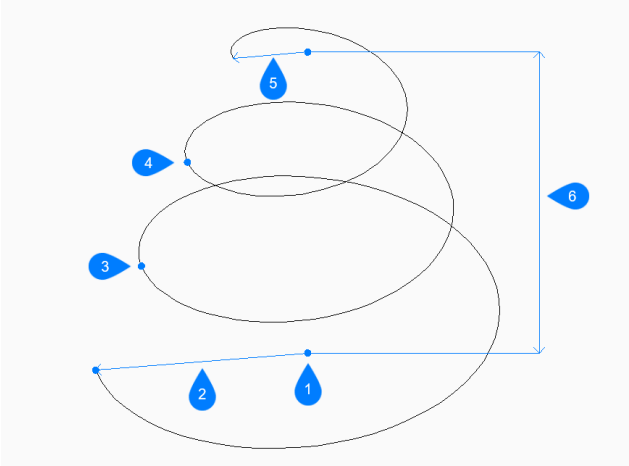
Creates a 2D spiral or a 3D helix.

Shape Lite Pro Mechanical BIM

Icon:

13.8.1 Description

Creates a 2D spiral or a 3D helix from a combination of options including center, radius, diameter, height, axis endpoints, turns, distance between turns, and twist.



1 Center of base

2 Base radius

3 Start turn 2

4 Start turn 3

5 Top radius

6 Height

13.8.2 Method

This command has 1 method to begin creating a helix:

- Specify center point of base

13.8.3 Options within the command

Specify center point of base
Allows you to begin creating a helix by specifying the center of the base.

Specify base radius
Specifies the radius of the helix's base.

Specify top radius
Specifies the radius of the helix's top.

Diameter
Specifies the diameter of the base or top of the helix.

Specify helix height
Specifies the height of the helix.

Axis endpoint
Specifies the axis endpoint to define height and orientation of the helix in 3D space. The center of the base is used as the other axis endpoint.

Turns
Specifies the helix's number of turns (revolutions).

Note: The number of turns cannot exceed 500.

turn Height
Specifies the distance between each revolution of the helix. The turn height and the number of turns define the height of the helix.

tWist
Specifies which direction the helix rotates.

- **CW**: clockwise
- **CCW**: counterclockwise

13.9 HELP command

Opens the BricsCAD Help Center.

Shape Lite Pro Mechanical BIM

Icon: ?

13.9.1 Description

Opens the BricsCAD Help Center Description to learn more about BricsCAD commands, system variables, and workflows. It opens in an external application window, using the default Web browser, allowing it

to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

13.10 HELPSEARCH command

Searches online help from the Command line.

Shape Lite Pro Mechanical BIM

13.10.1 Description

Searches the online BricsCAD help pages and then displays the result in your computer's default Web browser; works at the Command line.

13.11 HIDE command

Removes hidden lines from 3D entities.

Shape Lite Pro Mechanical BIM

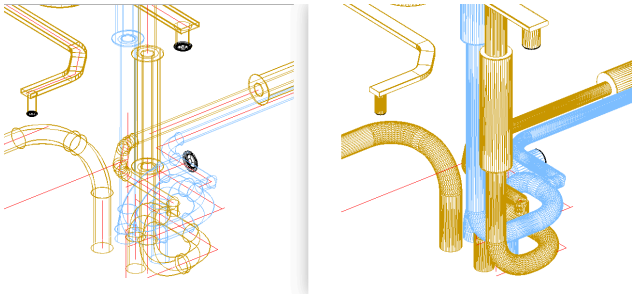
Icon:

Alias: HI

13.11.1 Method

This command offers only one method to remove hidden lines. The command is run automatically after typing “HIDE” in the Command line and hitting enter.

Displays no prompt in the Command line; the program immediately removes hidden lines.



Note: To return to wireframe display, change the visual style to 2D Wireframe.

13.12 HIDEOBJECTS command

Hides selected entities.

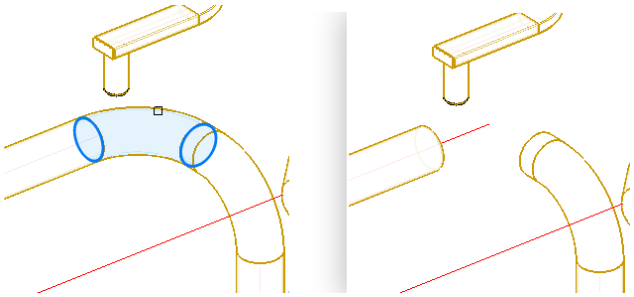
Shape Lite Pro Mechanical BIM

Icon:

13.12.1 Description

Hides selected entities, like using the Freeze option of the Layer command but for individual entities. Hidden entities optionally stay hidden between drawing sessions.

Note: To return hidden objects to view, use the UNISOLATEOBJECTS command.



Note: The OBJECTISOLATIONMODE system variable controls whether the hidden state is saved or not.

Note: The HIDEOBJECTS command is enabled under BEDIT sessions.

13.13 HYPERLINK command

Links files or web pages with entities in the drawing.

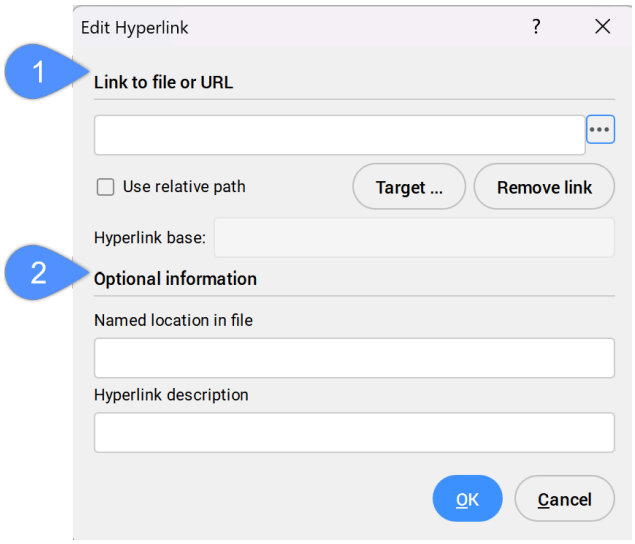
Shape Lite Pro Mechanical BIM

Icon:

13.13.1 Description

Associates various kinds of files or web pages to the selected entities through the **Edit Hyperlink** dialog box.

The **Edit Hyperlink** dialog box allows you to add or edit a hyperlink to a file or a web page.



1 Link to file or URL

2 Optional information

13.13.2 Link to file or URL

Specifies the link to a file or a URL to a web page.

- Click the Browse button (…) to open the **Select file** dialog box. Then, select the file and click **Open** to add the link to the **Link to file or URL** field.
- Type a URL to a web page in the **Link to file or URL** field.

Note: To remove the attached hyperlink, click the **Remove link** button.

Use relative path
If the file path in the **Link to file or URL** filed is absolute, the **Use relative path** option is greyed out.
You can manually shorten the file path of the attached hyperlink to make it relative. The **Use relative path** option becomes available and it is automatically ticked.

Hyperlink base
Displays the file path for all relative hyperlinks in the drawing.

Note: The base path is set by the **HYPERLINKBASE** system variable.

13.13.3 Optional information

Specifies additional information.

Named location in file
Specifies a bookmark name. The # character specifies locations (bookmarks) within files or web pages.

Hyperlink description
Adds an additional description of the hyperlink.

13.14 -HYPERLINK command

Links files or web pages with entities in the drawing via Command line.

Shape Lite Pro Mechanical BIM

13.14.1 Description

Associates various kinds of files or web pages to the selected entities through the Command line.

Note: This command is meant for use by macros and LISP routines.

13.14.2 Options within the command

Remove
Removes hyperlinks attached to the selected entities.

Insert
Adds a hyperlink to one or more selected entities.

Enter URL
Specifies the path file or the web page.

Enter named location
Specifies a bookmark name.

Note: The # character specifies locations (bookmarks) within files or web pages.

Enter description
Adds an additional description of the hyperlink.

Overwrite

Yes
Allows you to replace the existing hyperlink with a new one.

No
Leaves the existing hyperlink as it is and exits the command.

Note: The **Overwrite** option appears in the Command line only if the selected entity already has an attached hyperlink.

13.15 HYPERLINKOPTIONS command

Toggles the display of the hyperlink.

Shape Lite Pro Mechanical BIM

13.15.1 Description

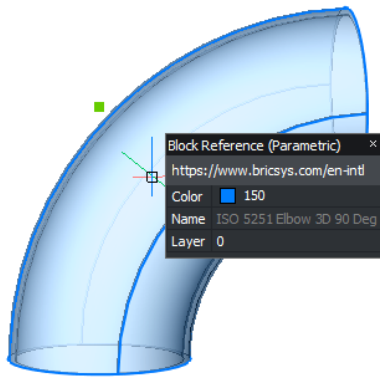
Toggles the display of the hyperlink cursor and URL tooltip and adds the **Hyperlink** option to the shortcut menu.

Note: The context menu can be accessed when the SHORTCUTMENU system variable is set to 4.

13.15.2 Options within the command

Display hyperlink cursor, tooltip and shortcut menu
Toggles the display of the "hyperlink" icon, tooltip displaying the URL, and the Hyperlink sub-menu on shortcut menus.

Note: The text of the tooltip is specified by **Hyperlink description** option of the HYPERLINK command.



14. I

14.1 ID command

Reports the x,y,z coordinate.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IDPOINT

14.1.1 Description

Reports the x,y,z coordinates of a specified point using the current coordinate system.

14.2 IEMBED command

Embeds an attached bitonal .tiff image file to the current drawing.

Shape Lite Pro Mechanical BIM

14.2.1 Method

Usually, the images attached to a drawing are saved in a separate file. This command saves a render image within the current drawing.

Note: To reverse this action use the IUNEMBED command.

14.3 IFCEXPORT command

Exports a BIM model to IFC.

Shape Lite Pro Mechanical BIM

Icon: 

14.3.1 Description

Exports a BIM model to IFC, partially or entirely. IFC attributes are managed dynamically, for example properties and property sets. This allows you to choose between IFC2x3 or IFC4 attributes.

When the IFCEXPORTVALIDATEMODEL system variable is enabled, an IFC model validation engine checks the exported IFC file to ensure that it is fully compatible with the official IFC2x3 and IFC4 specification rules by buildingSMART®. Any rule violation will be reported in the log file.

Note:

- The import and export of Alignments and TIN Surfaces with IFC4X1 is supported.
- A color resembling the material by layer color is applied for entities with Redway materials.
- To export properties with date values to IfcDate for IFC4, use the YYYY-MM-DD format. Different formats will be exported as IfcLabel.

The visibility rules for properties and property set can be changed via BIMPROPERTIES dialog box.

Note: Visibility rules are reflected in the exported IFC file.

The available export formats are:

- **IFC2x3 file** (*.ifc)
- **IFC4 Design Transfer View file** (*.ifc)
- **IFC4 Reference View file** (*.ifc)
- **IFC4x1 file (deprecated)** (*.ifc)
- **IFC4x3 file** (*.ifc)

Note:

- BricsCAD BIM is officially certified for **IFC4 Reference View** (Architecture).
- The **IFC4x3 file** (IFC 4.3.2.0) export format is available only when the **Experimental mode** is enabled.
- The **IFC4x1 file (deprecated)** will be removed in a future BricsCAD release.

14.3.2 Method

Select the entities to export or press **Enter** to export the entire model.

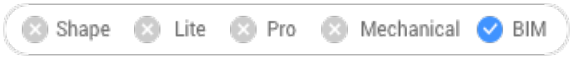
For more information about IFC export, see the **IFC import and export in BricsCAD** article.

Note:

- BricsCAD BIM created IFC files can be used in different coordination workflows with many tools (for example, Solibri).
- Some software allow for editing only IFC files created with specific Authoring Tools, while actively blocking files created with other tools. Other uses of BricsCAD BIM created IFC files on these software are possible (for example, viewing).
- When IFCEXPORTIDSPROPERTIESONLY system variable is ON, only the IFC and custom properties of the active namespace (selected in the drop-down menu) specified in the imported ISD file are exported.
- BricsCAD BIM also supports exporting files in RVT format (only for viewing).

14.4 IFCVALIDATE command

Checks the quality of an existing IFC file.



14.4.1 Description

Validates an existing IFC file before importing it.

14.4.2 Method

Opens the **Import file** dialog box to select the IFC file.

An IFC file validation report is displayed in the Command line, then you have to decide whether or not to import the IFC file.

Note: The report is also written to a log file next to the IFC file.

14.4.3 Options within the command

Yes
Imports the IFC file.

No
Does not import the IFC file.

14.5 IMAGE command

Opens the **Attachments** panel.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IM

14.5.1 Description

Opens the **Attachments** panel to display it in the current workspace. The **Attachments** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Attachments** panel can be either floating, docked or stacked.

14.6 IMAGEAPP command (Express Tools)

Specifies the image editor application used for the IMAGEEDIT command.

Shape Lite Pro Mechanical BIM

14.6.1 Method

Use the IMAGEAPP command to specify the image editor application, for example Microsoft Paint.

14.7 IMAGEADJUST command

Adjusts the properties of images through the **Properties** panel.

Shape Lite Pro Mechanical BIM

Alias: IAD

Keyboard Shortcuts: **click the image frame**

14.7.1 Method

After selecting one or more image entities by their frames the **Properties** panel will display the Raster Image properties.

The Image Adjust properties are:

Brightness
Lightens and darkens the image:

- 0 - very dark or black

- 50 - normal
- 100 - very light or white

Contrast
Increases and decreases the contrast::
- 0 - very low contrast
- 50 - normal
- 100 - very high contrast

Fade
Fades the image:
- 0 - no fade
- 100 - fully faded

14.8 -IMAGEATTACH command

Attaches raster images to the drawing via Command line.

Shape Lite Pro Mechanical BIM

14.8.1 Method

Introduce the path and file name of the image file, the insertion point, the scale factor and rotation factor to insert the raster image in the drawing.

14.8.2 Options within the command

Specifies how the program should remember the path to the image, which is saved in the drawing:

Full path
The full path of the image file is saved such as *D:\BricsCAD Training\EN\Exercises\Gearbox.png*.

Relative path
The path relative to the drawing folder is saved, such as *..\Exercises\Gearbox.png*.

No path
The path is not saved, and so the program searches for the image in the drawing folder or in support search file path that could be added with SRCHPATH command.

use Geocoding info
Uses geocoding data to determine the image's insertion point, scale, and rotation angle.

Geocoding file path
Specifies the name of a PGW file that contains the geocoding data.

Embedded geocoding info
Uses the geocoding information embedded in the image file.

Xy scales factors
Specifies the scale factors for the X and Y directions independently.

Rotation angle
Specifies the rotation angle.

14.9 IMAGEATTACH command

Attaches raster images to the drawing.

Shape Lite Pro Mechanical BIM

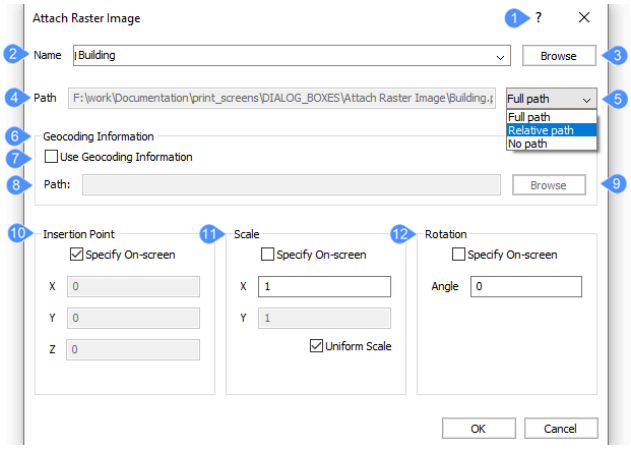
Icon:

Alias: IAT

14.9.1 Description

Opens the **Select Image File** dialog box to select an image file to reference into the current drawing. After selecting the file, click the **Open** button to display the **Attach Raster Image** dialog box. It enables you to specify where and how to attach the image.

The **Attach Raster Image** 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

14.9.2 Help

Opens the Bricsys Help article about the IMAGEATTACH command.

14.9.3 Name

Specifies the name of the file to attach.

14.9.4 Browse

Opens the **Select Image File** dialog box to allow you to choose a different raster file.

14.9.5 Path

Displays the path of the image file.

14.9.6 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*.

14.9.7 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.

14.9.8 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.

14.9.9 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.

14.9.10 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.

14.9.11 Angle

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.

14.10 IMAGECLIP command

Clips an image to a specified clipping boundary.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: ICL

14.10.1 Method

Select the image by clicking its frame and create a new clipping boundary.

Note:

- Clipped images can be edited directly through grips.
- This command can be entered transparently during commands ('imageclip).

14.10.2 Options within the command

ON
Turns On the clipping boundary that was turned Off by the **OFF** option.
The area outside the clipping boundary is hidden from the view.

OFf
Turns Off the clipping boundary so the selected image is fully visible again. The clipping boundary is saved, until it is deleted, and it can be turned on again with the **ON** option.

Invert
Inverts the clipping mode so that the image is clipped either outside or inside the specified boundary.

Note: The **Misc** section of the **Properties** panel also allows you to turn the clipping On and Off, and to invert it.

Delete boundary
Erases the clipping boundary from the selected image without prompting or warning you.

New boundary
Allows you to add a clipping boundary to all selected entities.

Delete old boundary(s)?
This option appears in Command line only when an existing boundary is detected.

Yes
Deletes the previous boundary(s).

No
Exits the command.

Select polyline
Allows you to choose any open or closed entity made from polylines, such as splines, polylines, rectangles, donuts, and polygons. While the polyline does not need to be closed, it cannot cross over itself.

Polygonal
Creates a non-rectangular clipping boundary, with a minimum of three sides.

Undo
Undoes the last polygonal segment.

Rectangular
Creates a rectangular clipping boundary.

14.11 IMAGEEDIT command (Express Tools)

Edits the selected image in an external image editor application.

Shape Lite Pro Mechanical BIM

Icon: 

14.11.1 Description

Opens the **Image Edit** dialog box that allows you to select an image to open in an external image editor application.

Note: The external image editor is specified with the IMAGEAPP command, for example Microsoft Paint.

14.12 IMAGEFRAME command

Toggles the IMAGEFRAME system variable.

Shape Lite Pro Mechanical BIM

Icon: 

14.12.1 Description

Toggle the IMAGEFRAME system variable to change the visibility and plotting for image frames. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'IMAGEFRAME.

- 0: sets the IMAGEFRAME system variable to 0.
- 1: sets the IMAGEFRAME system variable to 1.
- 2: sets the IMAGEFRAME system variable to 2.

14.13 IMAGEQUALITY command

Specifies the display quality of attached images.

Shape Lite Pro Mechanical BIM

Icon: 

14.13.1 Description

Specifies the display quality of attached images to increase performance or image resolution.

- Draft: increases performance by reducing color resolution, image size, and memory usage. This does not affect the quality of plotted images.
- High: increases image quality resulting in decreased performance for large images.

14.14 IMAGEOVERLAP command (Express Tools)

Changes the overlap distance for image tiling in superhatch.

Shape Lite Pro Mechanical BIM

14.14.1 Method

Enter the overlap distance for image tiling in superhatch.

14.15 -IMPORT command

Imports geometry from external files at the command prompts.

Shape Lite Pro Mechanical BIM

Note: This command opens all file formats supported by the IMPORT command.

14.15.1 Method

Specify the full path and file name of an importable file.

Note: Type ~ (tilde) to display the **Import file** dialog box, which allows you to select a file to import.

14.16 IMPORT command

Imports the geometry from external files into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IMP

14.16.1 Description

Opens the **Import file** dialog box to select a supported file type to import into the current drawing.

The supported file types to import are:

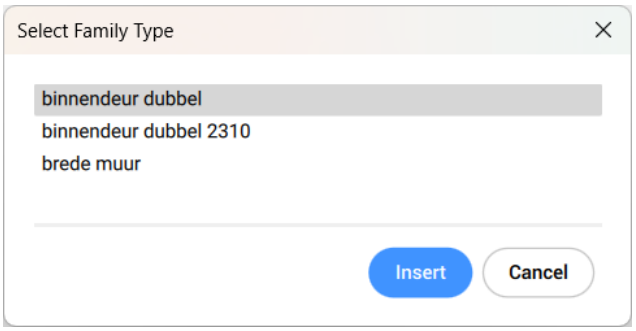
- **Drawing Exchange Format** (.dxf; .dwg)
- **Windows Metafile Formats*** (.wmf; .emf; .wmz; .emz)**(1) (2)**
- **Collada** (.dae)
- **Micro Station DGN File** (.dgn)

  Formats available with the BIM add-on:
- **Wavefront Object file** (.obj)
- **Rhino file** (.3dm)**(1) (2)**
- **SketchUp file** (.skp)**(1)**

  Note: The Trimble SketchUp files are imported as subdivision meshes (entities are imported as named blocks, render material definitions and the material mappings are imported on the meshes). The CONVTOSOLID and CONVTOMESH commands provide easy conversion between meshes and solids.
- **IFC file** (.ifc; .ifczip)

  Note: The **IFC Import Settings** dialog box opens.
- **Revit family** (.rfa)

  When a Revit® family contains multiple types, the **Select Family Type** dialog box displays, which enables you to import the preferred type of the Revit® family. The RFA files attributes are also imported.



- **Revit Project** (.rvt)

(1) Not available in BricsCAD for Linux.

(2) Not available in BricsCAD for macOS.

Note:

- BricsCAD currently supports the import of **Revit 2015-2024** 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.
- More 3D format files are available through a separate module, Communicator for BricsCAD®. You can download it from the Bricsys website.

  The import and export procedures using Communicator for BricsCAD® are controlled through a series of user preferences. See the **Communicator** section in the **Settings** dialog box.
- If the imported assembly has physical materials assigned to its parts, the materials can be imported together with the assembly structure. For each material in the imported file, a corresponding material will be created in the target document material library, and its name, density, specific heat and thermal conductivity will be copied from the source material. If the imported file contains several materials with the same name, the first one will be used. If the document already has a material with the same name in its material library, this material will be used instead.
- Materials copied to the document material library will not be erased with Undo operations.
- The IMPORT command supports components architecture based on regular blocks.

14.17 IMPRINT command

Imprints 2D entities onto planar faces of 3D solids and surfaces to create additional edges.

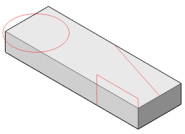
Shape Lite Pro Mechanical BIM

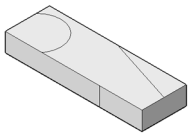
Icon:

Note: Use dynamic UCS (DUCS) to draw on the faces of 3D objects (UCSDETECT system variable =1).

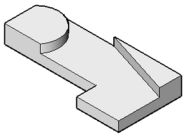
14.17.1 Method

Select a 3D solid that has at least one flat face, a surface or a region entity. Select the source objects, 2D entities lying on or intersecting a planar face of the selected entity and after each choose if delete or keep the source object.





With the entities imprinted on the solid, you can use commands like EXTRUDE or DMPUSHPULL to manipulate the newly created faces, as shown below.



14.18 INSERT command

Inserts blocks into the current drawing.

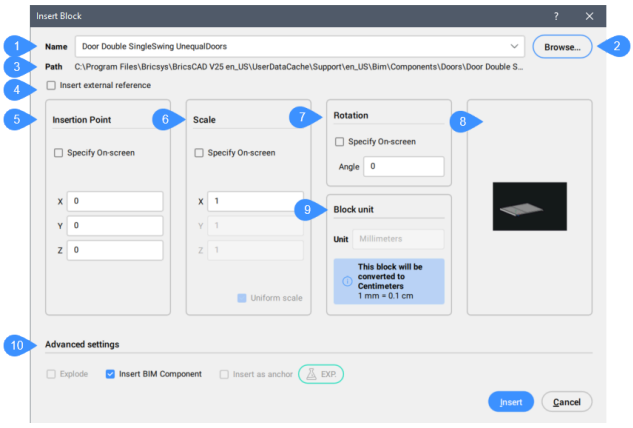
Shape Lite Pro Mechanical BIM

Icon:

Alias: DDINSERT, I

14.18.1 Description

Opens the **Insert Block** dialog box to insert a block instance from a block definition. The block definition may exist in the current drawing or as an external file.



1 Name

2 Browse

3 Path

4 Insert external reference

5 Insertion Point

6 Scale

7 Rotation

8 Block preview

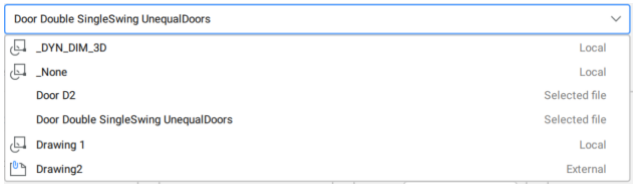
9 Block unit

10 Advanced settings

14.18.2 Name

Specifies the name of the block to insert, whose definition exists in the drawing, or the name of the file to insert as a block.

Note: Click the drop-down button to display the available blocks. It shows internal and external block types, with icons indicating each block's type.



14.18.3 Browse

Opens the **Select Drawing File** dialog box that allows you to select a DWG or DXF file from your computer or network.

14.18.4 Path

Displays the path to the file to insert as a block.

14.18.5 Insert external reference

Toggles whether the block is inserted as a local or external reference.

14.18.6 Insertion Point

Specifies the insertion point of the block in the drawing.

Specify On-screen
Allows you to dynamically select the insertion point of the block in the current drawing.

14.18.7 Scale

Scales the block.

Specify On-screen
Allows you to dynamically scale the block in the current drawing.

Input coordinates
Defines the scaling of the block.

X
Allows you to specify the scale factor along the X axis. Enter a negative value to mirror the block about the Y axis.

Y
Allows you to specify the scale factor along the Y axis. Enter a negative value to mirror the block about the X axis.

Z
Allows you to specify the scale factor along the Z axis.

Note:

- 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.

Uniform scale
Toggles whether the same scale factor is used for all axes.

Note: This option is enabled by default when inserting blocks created with the **Scale uniformity** option enabled (see the **BLOCK command** article).

14.18.8 Rotation

Specifies the rotation angle of the block about its insertion point, starting with the X-axis as 0 degrees.

Note: Positive angles rotate the block counter-clockwise and negative angles rotate the block clockwise.

Specify On-screen
Allows you to dynamically rotate the block in the current drawing.

14.18.9 Block preview

Displays a preview of the block.

14.18.10 Block unit

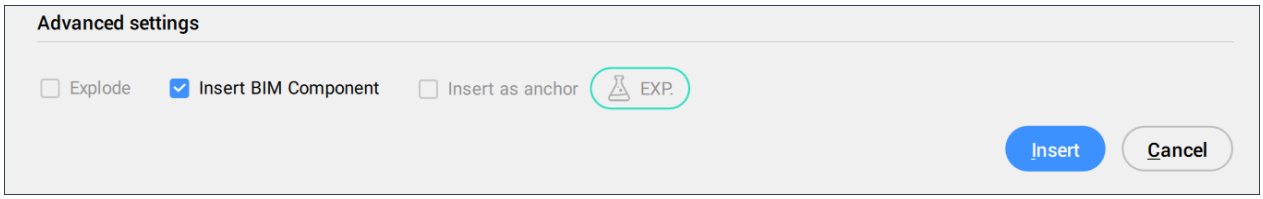
Controls the automatic scaling of the block with respect to the INSUNITS and INSUNITSSCALING system variables of the current drawing.

14.18.11 Advanced settings

Displays the available advanced settings, depending on the type of insert.

Explode
Toggles whether to explode the inserted block.

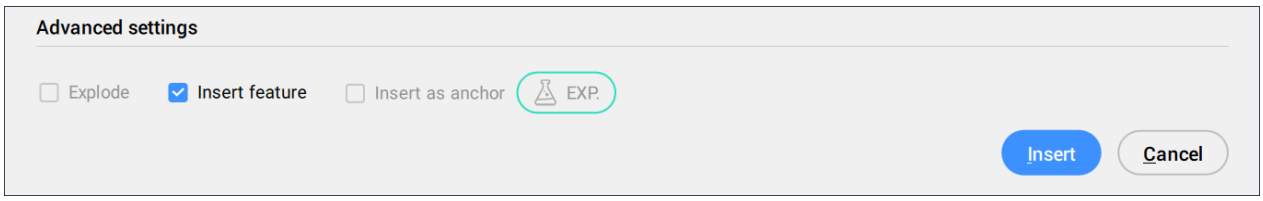
Insert BIM Component
Toggles whether to insert the block as a BIM Component.



Note:

- The **Insert BIM Component** option is available only if an external BIM component block is selected for insertion.
- This option is available for a BricsCAD BIM license. If you are using a BricsCAD Pro or a BricsCAD Mechanical license, the option becomes **Insert feature**.

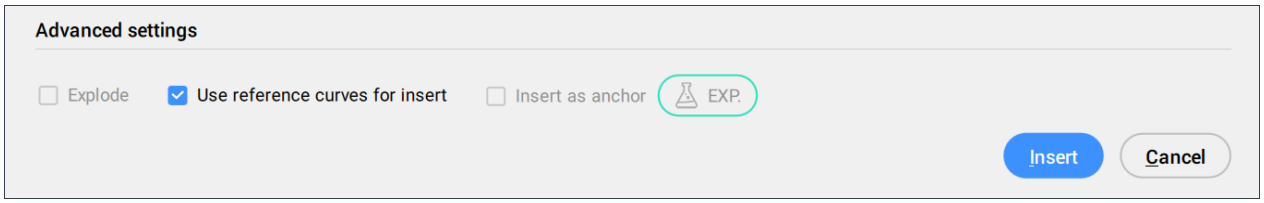
Insert feature
Toggles whether to insert the block as feature.



Note:

- The **Insert feature** option is available if an external mechanical component block is selected for insertion.
- BricsCAD Pro or higher is required for this option.

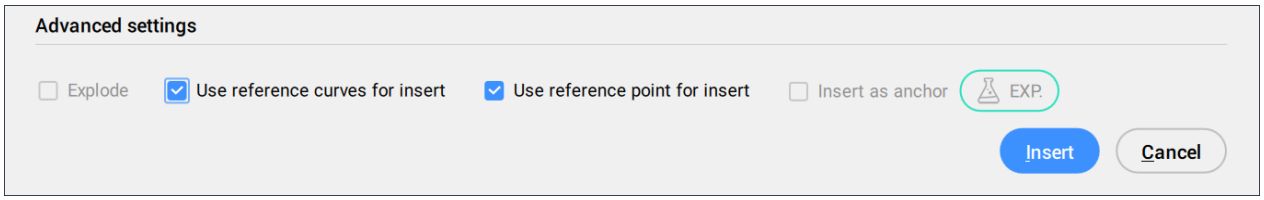
Use reference curves for insert
Toggles whether reference curves are used for insert.



Note:

- The **Use reference curves for insert** option is only available if the selected drawing contains reference curves. See also the **Fuzzy insert guided workflow** article.
- BricsCAD Pro or higher is required for this option.

Use reference point for insert
Toggles whether a reference point is used for insert.



Note:

- The **Use reference point for insert** option is only available if the selected DWG file contains a detail with guide curves. Details with guide curves can be created using the COPYGUIDED and MOVEGUIDED commands.
- It is available only when the **Use reference curves for insert** option is enabled. Otherwise, the option becomes gray out.
- BricsCAD Pro or higher is required for this option.

Insert Sheet Metal Form Feature
Toggles whether to insert sheet metal form feature.

Note:

- The **Insert Sheet Metal Form Feature** option is available only if a sheet metal form features block is selected for insertion and the Model Space contains SM flanges.
- This option is available for a BricsCAD Pro license or higher.

Insert as anchor
Anchors a block to a face of a solid. This anchor links the block to the face. The block will stay on the face when the host solid is moved or edited.

Note: BIMANCHOR is an experimental feature. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

14.18.12 Options within the command

When the **Insertion Point** > **Specify On-screen** option in the **Insert Block** dialog box is enabled, the following options appear in the Command line.

Edit inserted entity
Enables you to change the parameter expressions for the inserted entity. Continue editing individual parameters until you press **FINISH** to end the option. This option is also available in the Hot Key Assistant.

Enter parameter name
Specifies the name of the parameter.

Enter expression
Specifies the expression for the parameter.

FINISH
Finishes the editing of individual parameters.

SMART insert
Enables you to connect a Piping standard part to an existing Piping standard part. It automatically creates appropriate 3D constraints between the two parts and copies expressions for the parameters of the existing part to the new part. This option is also available in the **Hot Key Assistant**.

Rotate component
Allows you to change the rotation angle for the inserted entity.

set Base point
Allows you to change the base point for the inserted entity.

Note: By default is <0,0,0>.

Name
Allows you to change the component insert name.

insertion Type
Specifies the insertion type.

Local
Inserts the block as local component.

External
Inserts the block as external reference.

Flip
Allows you to flip the direction for the inserted entity.

Multiple
Allows you to you to insert multiple copies of the same entity by specifying an insertion point for each instance.

Note: Continue inserting entities until you press Enter to end the command.

Array
Allows you to create an associative array of the inserted entity by specifying the base point, distance between columns, distance between rows and end point of the array.

Direction
Allows you to select an existing axial entity to define the direction.

2Points
Selects two points to define the direction.

Zaxis
Selects the Z axis as direction.

Object
Allows you to select an axial entity.

Last
Uses the previous axis.

View
Allows you to specify a point on view direction.

Xaxis
Selects the X axis as direction.

Yaxis
Selects the Y axis as direction.

Single row
Distributes entity copies into a single row.

Rectangular
Distributes entity copies into any number of rows.

Columns
Specifies the number of the columns.

Rows
Specifies the number of the rows.

Placement
Sets the distance between the entities.

Accept
Accepts the resulting array.

Scale
Allows you to scale the inserted block.

Change target 3d solids
Allows you to apply the inserted entity to existing 3D solids in the current drawing.

Select target 3D solids
Selects the target 3D solids.

cleaR
Clears the selection set to ensure no solids are affected by the inserted entity.

Select all affected 3d solids
All solids intersecting or touching solids in the BC_SUBTRACT and BC_UNITE layers of the inserted entity are affected.

14.19 -INSERT command

Inserts blocks into the current drawing via Command line.

Shape Lite Pro Mechanical BIM

Alias: -I

14.19.1 Description

Inserts blocks through prompts in the Command line. It also prompts for attribute values, if any are part of the block definition.

14.19.2 Method

Specify the name of the block to insert or press Enter to accept the name of the previously inserted block and specify the insertion point, the scale factor, the rotation angle for the inserted block. BricsCAD prompts in the Command line the block inserted units, for example Units: Millimeters.

Note:

- The current drawing can be inserted as a block by typing the drawing name as block name.
- If the block contains attributes, the value of the ATTDIA (Attribute Dialog) system variable determines whether you are prompted in the Command line to set the attributes (ATTDIA=0) or through the **Edit Attributes** dialog box (ATTDIA=1).

14.19.3 Options within the command

? to list blocks in drawing

Lists the names of all blocks in the current drawing.
Enter part of a name to view the names of certain blocks. You can use wildcards like ? for an individual character and * for all characters.

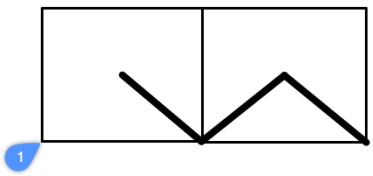
Note: If necessary, press F2 key to view the full list; see TEXTSCR command. You must restart the -INSERT command to insert the block.

~ to open the file dialog

Displays the **Insert Block** dialog box. Choose a DWG or DXF file and open it.

Select insertion point

Allows you to specify the X, Y and Z coordinates or pick a point for the block's insertion point (1). The Z coordinate is typically left at 0 for 2D blocks.



Corner

Indicates the size of the block by specifying a second corner of a rectangle. The insertion point is the first corner.

Note: Blocks can be edited directly through grips.

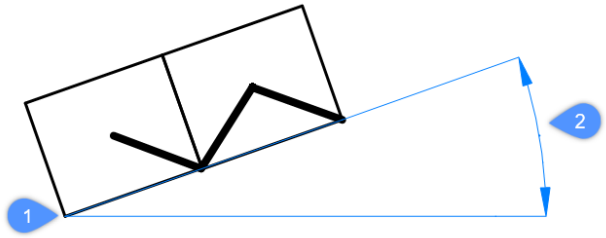
Edit inserted entity
Allows you to change the parameter expressions for the inserted entity.

FINISH
Finishes the editing of individual parameters.

SMART insert
Allows you to connect a Piping standard part to an existing Piping standard part. It automatically creates appropriate 3D constraints between the two parts and copies expressions for the parameters of the existing part to the new part.

Rotate component
Specifies the rotation angle (2) of the block about its insertion point (1), starting with the x-axis as 0 degrees:

- **Positive angles**: rotates the block counter-clockwise.
- **Negative angles**: rotates the block clockwise.



set Base point
Allows you to change the base point for the inserted entity.

Note: The default value is <0,0,0>.

insertion Type
Specifies the insertion type.

Local
Inserts the block as local component.

External
Inserts the block as external reference.

Flip
Allows you to flip the direction for the inserted entity.

Multiple
Allows you to insert multiple copies of the same entity by specifying an insertion point for each instance or creating an Array.

Note: Continue inserting entities until you press Enter to end the command.

Array
Allows you to create an associative array of the inserted entity by specifying the base point, distance between columns, distance between rows, and end point of the array.

Direction
Allows you to select an existing axial entity to define the direction.

2Points
Selects two points to define the direction.

Xaxis
Selects the X axis as direction.

Yaxis
Selects the Y axis as direction.

Single row
Distributes entity copies into a single row.

Rectangular
Distributes entity copies into any number of rows.

Base point
Allows you to specify a new base point for the array.

Columns
Specifies the number of the columns.

Rows
Specifies the number of the rows.

Placement
Sets the distance between columns.

Accept
Accepts the resulting array.

Change target 3d solids
Allows you to apply the inserted entity to existing 3D solids in the current drawing.

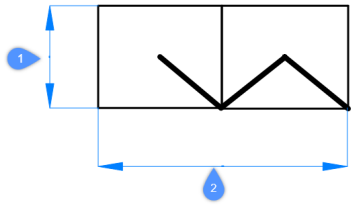
Select target 3D solids
Selects the target 3D solids.

cleaR
Clears the selection set to ensure no solids are affected by the inserted entity.

Select all affected 3d solids
All solids intersecting or touching solids in the BC_SUBTRACT and BC_UNITE layers of the inserted entity are affected.

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 scale
Specifies the scale factor along the X axis (2). Enter a negative value to mirror the block about the Y axis.

Y scale
Specifies the scale factor along the Y axis (1). Enter a negative value to mirror the block about the X axis.

Z scale
Specifies the scale factor along the Z axis.

14.20 INSERTALIGNED command

Inserts blocks aligned to entities.

Shape Lite Pro Mechanical BIM

Alias: INSAL

14.20.1 Description

Inserts blocks aligned with entities and optionally mirrors them. Hover the cursor over an existing entity in order to align the block to its geometry. Entity snap does not need to be on, but placement is more accurate when it is.

14.20.2 Options within the command

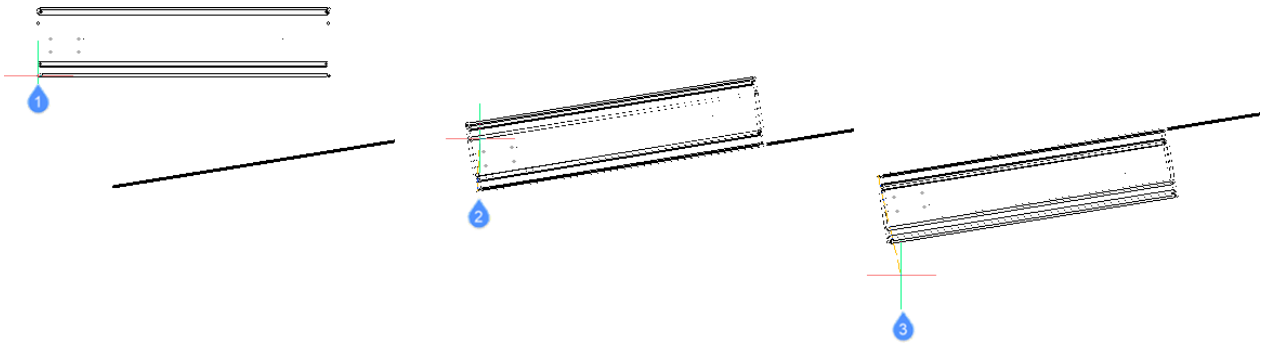
Block to insert

- Enter block name.
- ~ - opens a dialog box to choose an external drawing to insert.
- ? - lists the existing block definitions.

Block mirroring control point
Moves the cursor in order to flip the block about its insertion point.

X/Y/Z Scale
Sets the corresponding scale of the inserted block.

Multiple
Inserts multiple instances of the block.



1 Block being inserted

2 Block aligned with entity

3 Block mirrored about the entity

14.21 INSERTION command

Toggles the **Insertion** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



14.21.1 Description

Toggles the **Insertion** entity snap to enable or disable snapping to the insertion point of objects. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

14.22 INSERTLAYOUT command

Inserts a named layout of a drawing, as a Block Reference, in the current Paper Space.

Shape Lite Pro Mechanical BIM

14.22.1 Description

Select a drawing from the **Select drawing** dialog box and specify the layout name and the insertion point. A Block reference that replicates the entities in the layout will be inserted in the current Paper Space.

Note: The command is only available in Paper Space.

14.23 -INSERTLAYOUT command

Inserts a named layout of a drawing, as a Block Reference, in the current Paper Space.

Shape Lite Pro Mechanical BIM

14.23.1 Description

Specify the drawing path name to export the layout from and the layout name, together with the insertion point. A Block reference that replicates the entities in the layout will be inserted in the current Paper Space.

Note: The command is only available in Paper Space.

14.24 -INSERTMULTIPLELAYOUT command

Inserts multiple layouts, as Block References, in the current Paper Space via command line.

Shape Lite Pro Mechanical BIM

14.24.1 Description

Inserts named layouts of multiple drawings in the current Paper Space as Block References.

Note: The command is only available in Paper Space.

14.24.2 Method

Enter the margin to be used between viewports in the Paper Space.

Note: A positive value, or zero, is needed.

Enter the list of drawings to export layouts from, by specifying the drawing path names one by one. When finished, press Enter with an empty string to continue with exporting.

The -INSERTMULTIPLELAYOUT command now loops over the files and asks you to enter the layout name for each drawing.

Specify the base point and second point for the Block References.

14.25 INSERTOBJ command

Opens the **Insert object** dialog box.

Shape Lite Pro Mechanical BIM

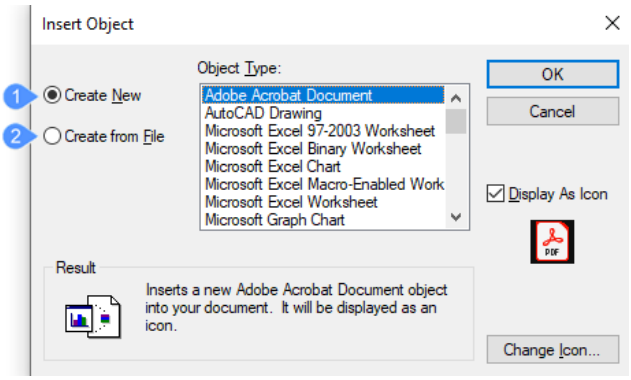
Alias: IO

Note: This is a Windows only command.

14.25.1 Description

Opens the **Insert object** dialog box to insert an OLE object into the current drawing.

The **Insert Object** dialog box allows you to insert a linked or embedded document in the current drawing.

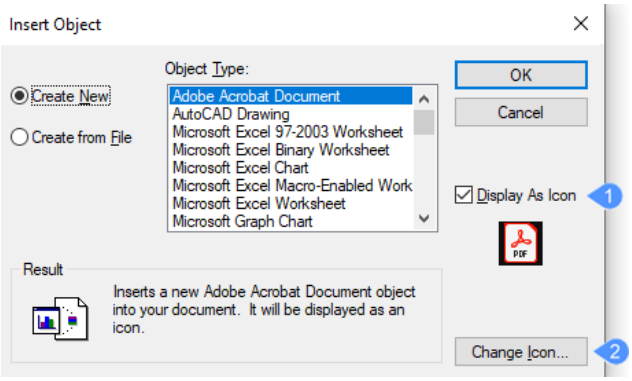


1 Create new Object

2 Create from File

Note: The object types listed varies according the software installed on your computer.

14.25.2 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. For more information, see the article **Change Icon dialog box**.

14.25.3 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.

14.26 INTERFERE command

Shows volumes and areas of interference between two sets of ACIS entities.

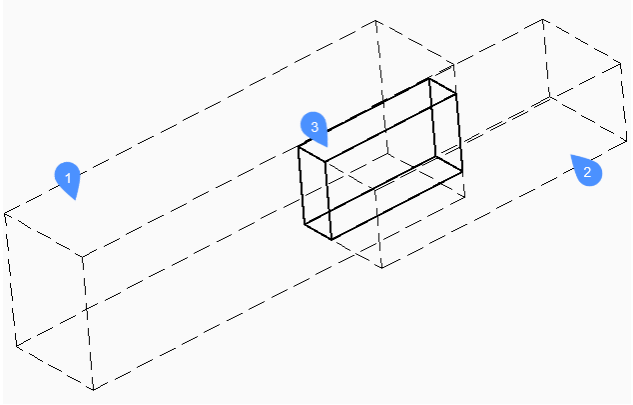
Shape Lite Pro Mechanical BIM

Icon:

Alias: INF

Note: This command can be entered transparently ('interfere) during the commands.

14.26.1 Method



Specifies the first (1) and the second (2) set of ACIS entities: choose one or more 3D solids and/or 2D regions.

You can respond with ALL to place all ACIS entities into a set and later use the Check first Set option to check them against each other. This avoids the need to create two sets of entities.

The entities in the first set are compared to the entities in the second set and results in an interference volume (3) or area which is created in a layer defined by the INTERFERELAYER system variable that is stored in the registry and has "Interferences" as an initial value.

Note: The Pro edition additionally shows volumes of interference between two sets of ACIS solids and then, optionally, creates new ACIS solids from the common portions of pairs of intersecting solids, placing them on layer "Interferences".

The interference solid(s) remain(s) in the drawing when the command ends.

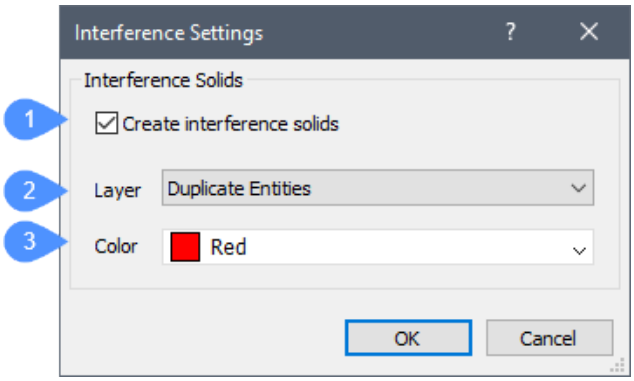
Note: Entities which are on the layer that is set by the INTERFERELAYER system variable are not accepted when selecting entities, as it is considered the output layer only.

Note: When you select an interference entity in the model or in the **Structure** panel, the originating entities highlight along with the selected interference entity.

14.26.2 Options within the command

Nested selection
Selects ACIS entities inside blocks or xrefs.

Settings
Displays a dialog box:



1 **Create interference solids**: when ticked, new solids are created from the interference area or volume.

2 **Layer**: specifies the layer on which interference entities are created. Default = layer stored in INTERFERELAYER system variable. Do not select a layer of selected entities as it is considered the output layer only. It is recommended to select an empty layer or accept the default layer.

3 **Color**: specifies the color of the interference entities; the color of the Interference layer is changed to match this color.

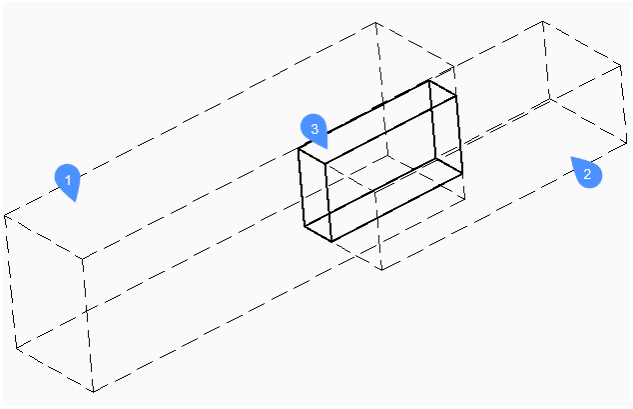
checK first set
Checks the interference of entities in the first set, when it contains two or more entities, and then creates interference entity(ies) of them.

14.27 -INTERFERE command

Shows volumes and areas of interference between two sets of ACIS entities.

Shape Lite Pro Mechanical BIM

14.27.1 Method



Specifies the first (1) and the second (2) set of ACIS entities: choose one or more 3D solids and/or 2D regions.

You can respond with ALL to place all ACIS entities into a set and later use the Check first Set option to check them against each other. This obviates the need to create two sets of entities.

The entities in the first set are compared with the entities in the second set and results an interference volume (3) or area which is created in layer defined by INTERFERELAYER system variable that is stored in the registry and has initial value for "Interferences".

Note: The Pro edition additionally shows volumes of interference between two sets of ACIS solids and then, optionally, creates new ACIS solids from the common portions of pairs of intersecting solids, placing them on layer "Interferences".

The interference solid(s) remain(s) in the drawing when the command ends.

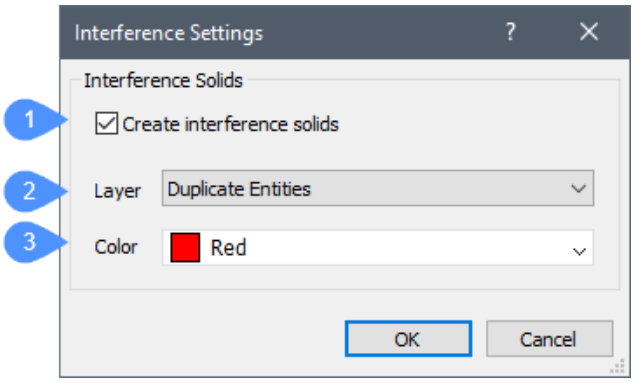
Note: Entities which are on the layer that is set by the INTERFERELAYER system variable are not accepted when selecting entities, as it is considered the output layer only.

Note: When you select an interference entity in the model or in the **Structure** panel, the originating entities highlight along with the selected interference entity.

14.27.2 Options within the command

Nested selection
Selects ACIS entities inside blocks or xrefs.

Settings
Displays a dialog box:



1 **Create interference solids**: when ticked, it makes new solids from the interference area or volume.

2 **Layer**: specifies the layer on which interference entities are created. Default = layer stored in INTERFERELAYER system variable. Do not select a layer of selected entities as it is considered the output layer only. It is recommended to select an empty layer or accept the default layer.

3 **Color**: specifies the color of the interference entities; the color of the Interference layer is changed to match this color.

checK first set
Checks the interference of entities in the first set, when it contains two or more entities, and then creates interference entity(ies) of them.

Create interference solids?
Allows you to choose whether to create interference solid or not.

14.28 INTERFERENCEMANAGERPANELOPEN command (Experimental)

Opens the **Interference Manager** panel.

Shape Lite Pro Mechanical BIM

14.28.1 Disclaimer

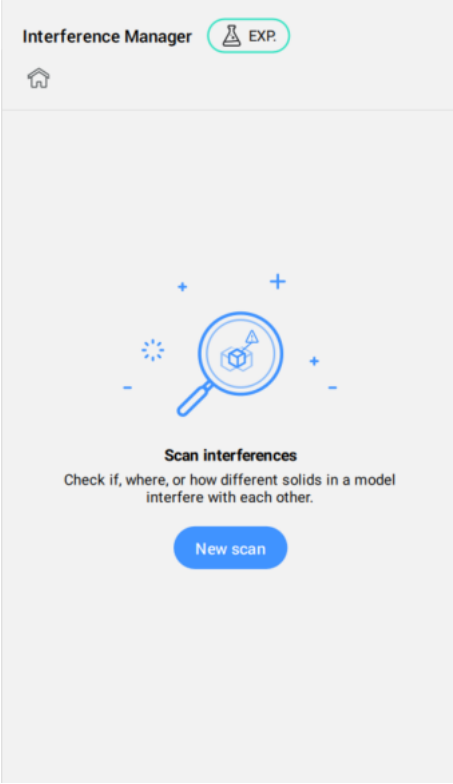
EXP.

The **Interference Manager** is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

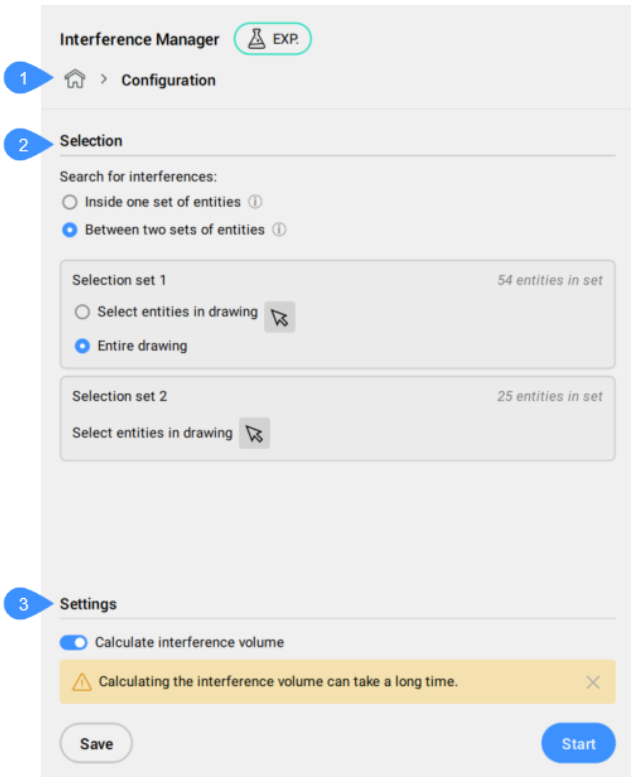
The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

14.28.2 Description

Opens the **Interference Manager** panel to identify and manage interference between 3D solids or block references in complex 3D models.



Press the **New scan** button to open the **Configuration** window that allows you to start scanning interferences.



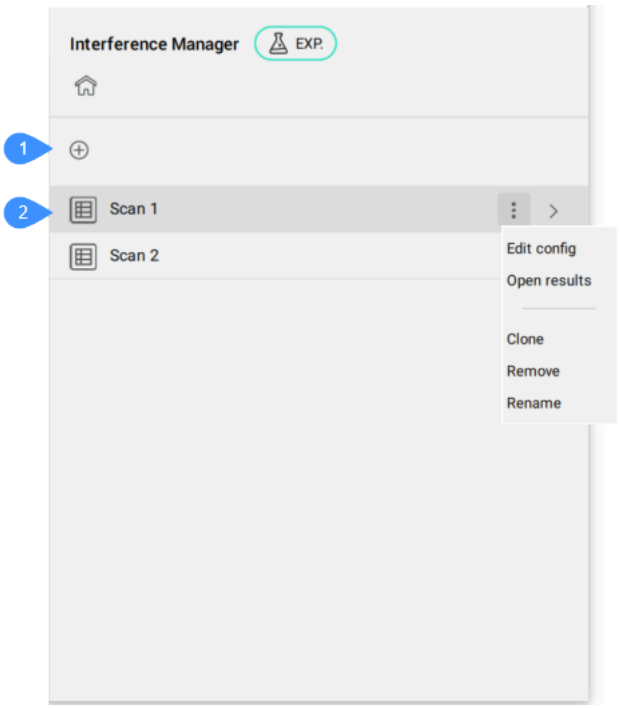
1 Home

2 Selection

3 Settings

14.28.3 Home

Restarts the interference scanning or opens the **Home** window that contains the saved scans.



1 New scan

2 List of saved scans

New scan
Opens the **Configuration** window to start a new scan.

List of saved scans
Lists the saved scans.
Right-click a saved scan to open a context menu with the following options.

Edit config
Opens the **Configuration** window to edit the scan.

Open results
Opens the **Result** window for the scan.

Clone
Creates a clone of the scan.

Remove
Removes the scan from the list.

Rename
Renames the scan.

14.28.4 Selection

Defines the entities that will be searched for interferences.

Inside one set of entities
Searches for interference in a specific set of entities.

Between two sets of entities
Searches for interference between two distinct sets of entities.

Selection set
Defines the entity selection set.

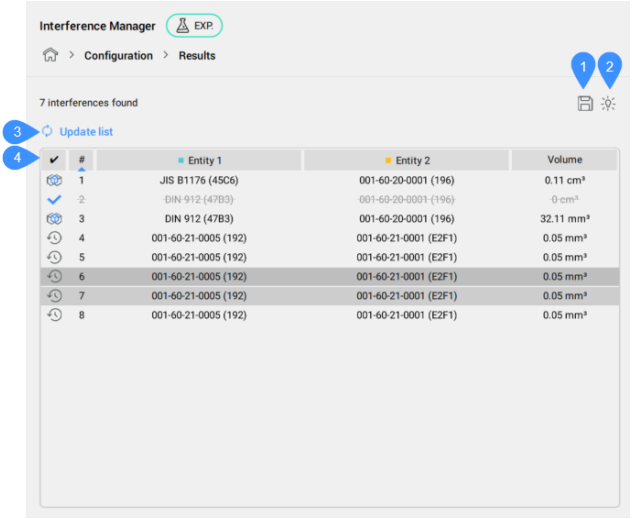
Select entities in the drawing
Allows you to select the entities directly from the drawing area.

Entire drawing
Selects all entities in the drawing.

14.28.5 Settings

Calculate interference volume
Toggles the option to calculate the volume of interference.

14.28.6 Results



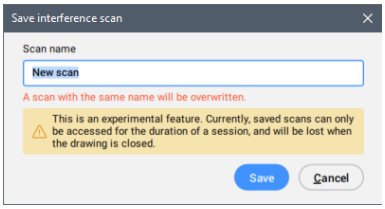
1 Save

2 Highlight the interfering entities

3 Update list

4 List of the entities that interfere

Save
Saves the configuration and results of this scan. The **Save interference scan** dialog box opens.



Note: Saved scans can only be accessed during the current session and will be lost when the drawing is closed.

Highlight the interfering entities
Toggles the display of the interfering entities and the interference volumes in distinct colors, while the rest of the entities are transparent. Interfering entities are colored in cyan and yellow, while the interference volume is red.

Update list
Re-evaluates all outdated interference pairs in the result lists.

List of the entities that interfere
Lists all interference pairs found between entities and displays the interference volumes.
The list is sortable by any of its columns. Easily navigate through interference pairs using the up and down keys. Select one or multiple rows to visualize their interferences simultaneously on the model.
While keeping the **Results** window open as a checklist, you can execute commands, such as DMPUSH-PULL or MANIPULATE to resolve geometric interference.

Status
Displays the status of the interfering entities.

Interference
Entities are interfering.

Outdated
One of the interfering entities might be updated.

Resolved
Interference between entities has been resolved.

Cannot be found
One of the interfering entities cannot be found. It might be deleted or unloaded.

Position
Displays the position number of the pair of interfering entities.

Entity 1
Displays the first entity in the pair of interfering entities. It is colored cyan in the model.

Entity 2
Displays the second entity in the pair of interfering entities. It is colored yellow in the model.

Volume
Displays the value of the interference volume. The interference volume is colored red in the model.

14.29 INTERFERENCEMANAGERPANELCLOSE command (Experimental)

Closes the **Interference Manager** panel.

Shape Lite Pro Mechanical BIM

14.29.1 Disclaimer

EXP.

The **Interference Manager** is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

14.29.2 Description

Closes the **Interference Manager** panel to hide it from the current workspace. If the **Interference Manager** panel is stacked when you close it, the **Interference Manager** tab or icon is removed from the stack.

14.30 INTERSECT command

Performs Boolean intersection operations on 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icon:

Alias: IN

Note: In the BricsCAD Lite license level, the command applies to region entities only.

14.30.1 Description

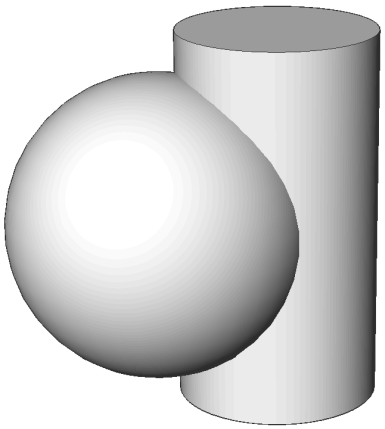
Performs Boolean intersection operations on 3D solids and 2D regions by removing all except portions in common.

Note: If solids and regions do not intersect, BricsCAD erases them.

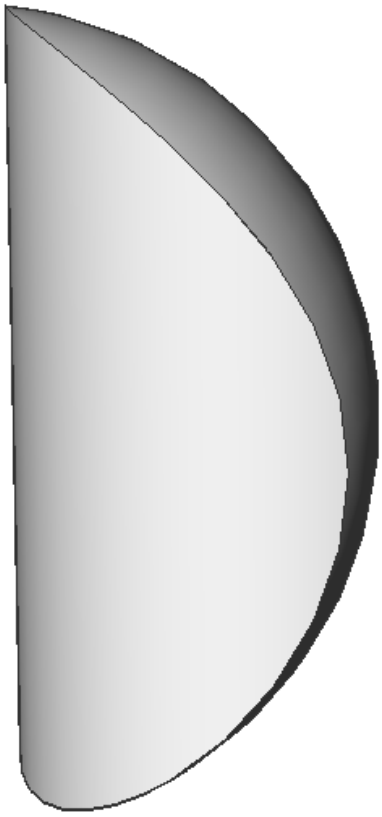
14.30.2 Method

Specify the 3D solids and/or 2D regions to be intersected. The program removes all except the volumes and areas in common from the selected entities.

The entities selected:



Result:



Note: You can interrupt the command by pressing the CANCEL button.

14.31 INTERSECTION command

Toggles the **Intersection** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



14.31.1 Description

Toggles the **Intersection** entity snap to enable or disable snapping to intersection. You can launch this command at the Command line to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

14.32 ISAVEAS command

Saves images.

Shape Lite Pro Mechanical BIM

14.32.1 Description

Saves images to your computer through the **Save Image File** dialog box.

14.33 ISOLATEOBJECTS command

Hides all but the entity(ies) selected.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: ISOLATE

14.33.1 Description

Only selected entities are displayed. Other entities are hidden.

Note: Use the UNISOLATEOBJECTS command to make the hidden entities visible again.

Note: The OBJECTISOLATIONMODE variable controls what happens with the hidden state of the entities that weren't selected to be hidden.

Note: The ISOLATEOBJECTS command is enabled under BEDIT sessions.

14.34 ISOPLANE command

Toggles the SNAPISOPAIR system variable.

Shape Lite Pro Mechanical BIM

Alias: IS

14.34.1 Description

Toggles the SNAPISOPAIR system variable to specify the drafting plane for isometric drawings. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'ISOPLANE.

- Left: sets the SNAPISOPAIR system variable to Left.
- Right: sets the SNAPISOPAIR system variable to Right.
- Top: sets the SNAPISOPAIR system variable to Top.
- Toggle: changes the SNAPISOPAIR system variable to the next setting. It cycles in order from the current setting (left-top-right-left).

14.35 IUNEMBED command

Converts an embedded image to an attached one.

Shape Lite Pro Mechanical BIM

14.35.1 Description

Converts an embedded image into an attached image by exporting the embedded image to a file (which the image entity is then attached to).

Note: This command reverts the action of IEMBED command.

15. J

15.1 JOIN command

Joins 2D entities at their common endpoints.

Shape Lite Pro Mechanical BIM

Icon: 

15.1.1 Description

Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints.

Note: The type of the resulting entity depends on the types of the input entities and on their coplanarity.

15.1.2 Method

Two 2D polylines are joined into a single:

- 2D polyline- when they are coplanar.
- 3D polyline- when they are not coplanar and if they are composed only of straight segments.
- Spline- when they are not coplanar and if at least one of the polylines has a bulged (arc) segment.

A 2D polyline and a 3D polyline are joined into a single:

- 2D polyline: when they are coplanar.
- 3D polyline: when they are not coplanar and if the 2D polyline only has straight segments.
- Spline: when they are not coplanar and if the 2D polyline has at least one arc segment.

A line and a circular arc (or a polyline with arcs) are joined into a single:

- 2D polyline: when they are coplanar.
- Spline: when they are not coplanar.

The following pairs of entities are joined into a spline:

- Line and an elliptical arc.
- Spline and another open entity, such as an elliptical arc or polyline.
- Helix and another open entity, such as a line or arc.

Even when there are gaps between these entities, the command joins them into a single entity:

- Collinear lines: joined into a single line.
- Coplanar arcs (with same radii and center points): joined into an arc or a circle.
- Coplanar elliptical arcs (with same major and minor axes): joined into an elliptical arc or ellipse.

Circular and elliptical arcs are joined in a counterclockwise direction starting from the source arc.

Note: Collinear means the entities lie in the same imaginary line. Coplanar means the entities lie in the same plane.

16. K

16.1 KEEPME command

Adds changed entities to the source drawing, when two somewhat different drawings are being compared visually.

Shape Lite Pro Mechanical BIM

Icon: ✓

Note: The command works only during a session begun by the DWGCOMPARE command.

16.1.1 Method

Select one or more entities or else type ALL to select all entities in the drawing to add them to the source drawing.

17. L

17.1 LANDXMLEXPORT command

Exports Civil entities to a LandXML file.

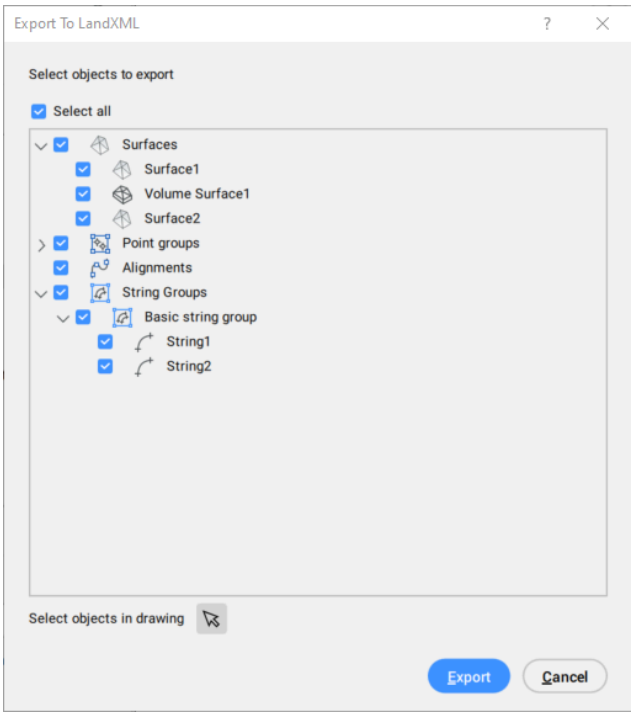
Shape | Lite | Pro | Mechanical | BIM

Icon: LAND XML

17.1.1 Description

Allows you to select the BricsCAD Civil entities (Civil points, TIN surfaces, Horizontal Alignments and 3D Alignments, Strings) you want to export to a LandXML file.

Opens the **Export To LandXML** dialog box.



17.1.2 Select all

When checked, all entities in the drawing are selected for export.

17.1.3 Select objects in drawing

Click the select icon ( ) to temporarily dismiss the **Export To LandXML** dialog box and select entities in the drawing to export.

17.1.4 Export

Opens the **Save LandXML file** dialog box where you can specify the location and name of the output LandXML file.

17.2 -LANDXMLEXPORT command

Exports Civil entities to a LandXML file via Command line

Shape Lite Pro Mechanical BIM

17.2.1 Method

Select the entities you want to export.

Opens the **Save LandXML file** dialog box where you can specify the location and name of the output LandXML file.

Note: Exports BricsCAD Civil entities, such as Civil points, TIN surfaces, Horizontal Alignments, 3D Alignments, and Strings.

17.3 LANDXMLIMPORT command

Creates TIN surfaces or alignments from a LandXML file.

Shape Lite Pro Mechanical BIM

Icon: 

17.3.1 Description

Opens the **Open LandXML file** dialog box to select an XML file to import.

The following BricsCAD Civil entities can be imported from the specified LandXML file: Civil Points, TIN Surfaces, Horizontal Alignments (both Alignments by PI and Alignments by Elements are supported), 3D Alignments, and Strings.

17.3.2 Options within the command

Draw Breaklines as Polylines
Specifies if breaklines are imported as 3D polylines.

Note: This option is available when importing TIN surfaces with breaklines.

The units in the drawing are different than those in the XML file. What would you like to do:
This option is available if the units in the current drawing do not match the units in the input XML file.

Scale
Scales the XML file units.

Import without scaling
Imports the XML file without scaling.

Cancel
Cancels the command.

17.4 LAYCUR command

Moves selected entities to the current layer.

Shape Lite Pro Mechanical BIM

Icon:

17.4.1 Description

Moves selected entities to the current layer without having to specify the name of the current layer.

You are prompted:

Select entities to be moved to the current layer

Select one or more entities you want to move to the current layer. You can continue selecting entities until you press Enter to end the command. The Command line indicates how many entities were moved and to what layer:

 # entities were moved to the current layer ("LayerName1").

17.5 -LAYDEL command (Express Tools)

Permanently deletes a layer from the current drawing, with all entities on it, via Command line.

Shape Lite Pro Mechanical BIM

17.5.1 Method

Select an entity on the layer to be deleted. All the entities on that layer are deleted, along with the layer.

17.5.2 Options within the command

Type-it
Allows you to enter the layer name.

Name
Allows you to enter the layer name.

?
Allows you to enter the layer name(s) to list.

*****
Lists all available layers in the Command line and Prompt History window.

Undo
Clears the previous selection.

done
Lists in the Command line and Prompt History window the layer(s) that will be deleted.

Yes
Permanently deletes the layer with all entities on it.

No
Exits the operation.

17.6 LAYDEL command (Express Tools)

Permanently deletes a layer from the current drawing, with all entities on it.

Shape Lite Pro Mechanical BIM

Icon:

17.6.1 Method

Select an entity on the layer to be deleted. All the entities on that layer are deleted, along with the layer.

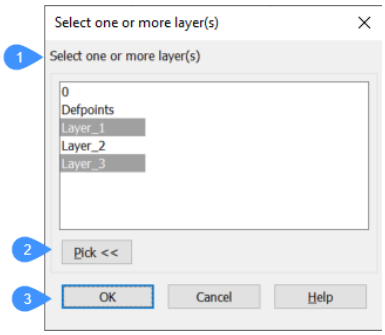
17.6.2 Options within the command

Type-it
Allows you to enter the layer name.

List
Allows you to enter the layer name(s) to list.

*
Lists all available layers.

Name
Opens the **Select one or more layer(s)** dialog box to select one or more layers to delete.



1 Select one or more layer(s)

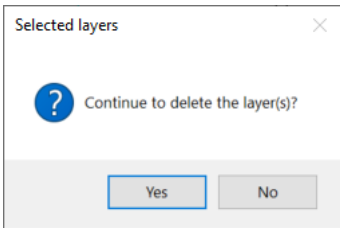
2 Pick<<

3 OK

Select one or more layer(s)
Allows you to select one or more layers to delete.

Pick<<
Allows you to select entities in the drawing which layers will be deleted.

OK
Opens the **Selected layers** dialog box, which allows you to choose whether to delete the layers.



Continue to delete the layer(s)?

Yes
Permanently deletes the layer with all entities on it.

No
Exits the operation.

17.7 -LAYER command

Manages layers through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -LA

17.7.1 Description

Creates layers and layer states and changes their properties.

Note: The current layer cannot be turned off and frozen.

17.7.2 Options within the command

?
Lists the names of layers in the current drawing.

New layer
Creates new layers.

Note: You can create several new layers by separating each name with a comma (,).

Make new current layer
Creates a new layer, and then makes it current.

Set layer as current
Sets a layer as the current one.

Rename
Renames the layer.

Color
Changes the color of layers.

Linetype
Changes the linetype of layers.

LineWeight
Changes the lineweight of layers.

TRansparency
Sets the entity transparency factor between 0 (fully opaque) and 90 (fully transparent).

MATerial
Sets the material property for all entities on the layer.

Plot
Changes the plot status of layers.

stAte
Manages layer states.

Save
Saves the current layer state.

Restore
Restores a saved layer state.

Edit
Edits a saved layer state.

reName
Renames a saved layer state.

Delete
Deletes a saved layer state.

Import
Displays the Import Layers States dialog box, which allows you to import a layer state from a file into the current drawing.

eXport
Exports a saved layer state in the current drawing to a layer state file.

turn layer ON
Turns layers on that were previously turned off.

turn layer OFf
Turns layers off to hide their entities from view.

Freeze
Freezes layers so that they are hidden from view.

Thaw
Thaws layers that were previously hidden with the Freeze option.

LOck
Locks layers so that their entities remain visible but cannot be edited.

Unlock
Unlocks layers that were previously locked.

Description
Allows you to set the description property value of existing layers.
Enter the description, then enter the name(s) of the layer(s) to apply the description to. When listing multiple layer names, separate the names with a comma (,).
A warning prompt is displayed when you list a layer with an existing description.

rEconcile
Sets the unreconciled property of one or more unreconciled layers. When listing multiple layer names, separate the names with a comma (,).

Xref
Resets the XRef layer property override in the current drawing to the layer settings in the XRef.

Note: The VISRETAIN system variable:

Controls the visibility, color, linetype and lineweight of an XRef, and if path changes to nested XRefs are saved. If the PSTYLEPOLICY system variable is off (0), also controls the plotstyles of XRef-dependent layers.

- If Off (0): Changes made to XRefs-dependent layers in the current drawing are valid in the current session only and are not saved with the drawing. When the current drawing is reopened, the layer table is reloaded from the reference drawing and the current drawing reflects those settings. The layer settings affected are On, Off, Freeze, Thaw, Color, Ltype, LWeight, and PStyle (if PSTYLEPOLICY is set to 0).
- If On (1): Layer settings are saved with the current drawing's layer table and persist from session to session.

17.8 LAYER command

Opens the **Layers** panel.

Shape Lite Pro Mechanical BIM

Alias: DDLMODES, LA

17.8.1 Description

Opens the **Layers** panel to display it in the current workspace. The **Layers** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Layers** panel can be either floating, docked or stacked.

17.9 LAYERP command

Restores layer properties to their previous state.

Shape Lite Pro Mechanical BIM

Icon: 

17.9.1 Description

Restores layer properties to their previous state one change at a time. This command only works when the LAYERPMODE system variable is enabled.

There are no prompts. The Command line indicates: Previous layer settings were restored.

17.10 LAYERSPANELCLOSE command

Closes the **Layers** panel.

Shape Lite Pro Mechanical BIM

17.10.1 Description

Closes the **Layers** panel to hide it from the current workspace. If the **Layers** panel is stacked when you close it, the **Layers** tab or icon is removed from the stack.

17.11 LAYERSPANELOPEN command

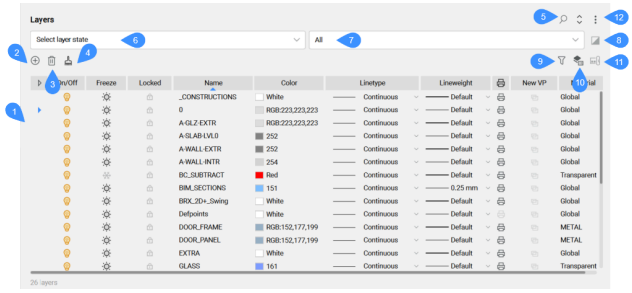
Opens the **Layers** panel.

Shape Lite Pro Mechanical BIM

17.11.1 Description

Opens the **Layers** panel link to display it in the current workspace. The **Layers** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Layers** panel can be either floating, docked or stacked.

The **Layers** panel allows you to edit layer names and properties in the current drawing, and to create, delete, edit, switch layers, layer states and layer filters.



1 Layer list

2 Add layer

3 Delete

4 Purge

5 Search

6 Layer State Control

7 Filter used

8 Invert filter

9 Layer Explorer

10 Layer States

11 Hide xref symbols on/off

12 Menu

17.11.2 Layer list

Displays a list of layers and their properties. The list can 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 can sort the layer list in order or reverse order for any of the properties by clicking once or twice on the column header.

A context menu displays when you right click menu on a layer name.

New
Creates a new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

Set Current
Specifies the current working layer

Rename
Renames the selected layer

Delete
Deletes the selected layer

Purge
Purges layers from the current drawing. You cannot purge the following layers:

- The layer 0.
- The current layer.
- Any layer with entities.

Merge to
Merges the selected layers to a target layer.

Note: Merged layers will be purged from the drawing.

Bind
Binds the selected layer(s) from Xrefs to the drawing.

Select All
Selects all available layers. You can also use the keyboard shortcut Ctrl+A.

Clear All
Deselects all selected layers.

Select All but Current
Selects all available layers except the current layer.

Invert Selection
Inverts the selected layers.

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.

Indicate Layers in Use
If the option is active, it displays a check mark in the **Current** column:

- **Used**: the layer is used in the drawing.
- **Unused**: the layer is not used in the drawing.

Right-click on the column headers to customize them. You can enable or disable the columns and restore the column positions.

Index
Specifies the layer's index.

Current

Specifies the current working layer. A check mark ( ) appears for the current layer.

Note: Only one layer can be current at a time.

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 light bulb icon for one of the selected layers. The status is applied to all selected layers.

- **On** ( ): displays entities that are on that layer.
- **Off** ( ): hides entities that are on that layer.

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 freeze 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.

Note: Freezing and thawing layers is similar to turning them On and Off. However, when working with drawings with many layers, freezing unnecessary layers helps speed up display and regeneration. Entities on a frozen layer are not taken into account during a zoom to extents.

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.

Name

Specifies the layer name. To rename a layer, double-click the layer name or select **Rename** from the right-click menu.

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.

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 **Color** dialog box is displayed. The color you select is applied to all selected layers.

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.

Lineweight

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.

Transparency

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.

Note: The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

Plot style

Specifies the plot style of the layers. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only. If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The **Select Plot Style** dialog box is displayed. The plot style you select is applied to all selected layers.

Plot

Specifies the print status of the layers. Entities on **No Print** layers remain visible in the drawing but are not printed. To change the status, select one or more layers, and then click the printer icon for one of the selected layers. The status is applied to all selected layers.

- **Print** (the printer icon is highlighted): Prints the layer.
- **No Print** (the printer icon is not highlighted): Does not print the layer.

Some layers created by BricsCAD, such as Defpoints, are set to **No Print** by default and cannot be changed.

New VP

Specifies the default Thaw/Freeze status of layers for new layout viewports. To change the status, select one or more layers and then click the VP icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw** (the VP icon is not highlighted): displays entities that are on that layer when a new viewport is created.
- **Freeze** (the VP icon is highlighted): hides entities that are on that layer when a new viewport is created.

Material

Specifies the material of the layers. To change the material, select one or more layers and then double-click the material for one of the selected layers. The material you select is applied to all selected layers. 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.

VP Freeze

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 VP Freeze 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** ( ): hides entities that are on that layer in the current layout or viewport.

VP Color

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 **Color** dialog box is displayed. The color you select is applied to all selected layers.

VP Linetype

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.

VP Lineweight

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.

VP Transparency

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.

Note: The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

VP Plot Style

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.

17.11.3 Add layer

Creates new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

17.11.4 Delete

Deletes layers, except for the layer with the name **0** and any layer with entities, which cannot be deleted.

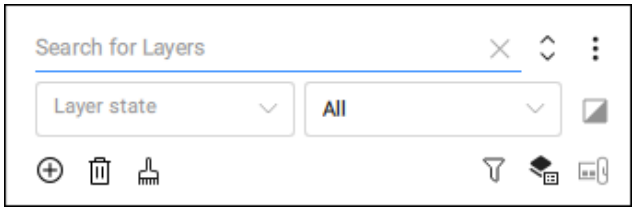
17.11.5 Purge

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.

17.11.6 Search

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.



Note: You can remove the active filter by pressing the close button on the right side of the search box.

17.11.7 Layer State Control

Displays the current layer state if there are any defined in the drawing. Click the down-arrow button to choose a different layer state.

17.11.8 Filter used

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.

17.11.9 Invert filter

Inverts the content of the current filter so that the panel displays all layers not included in the filter.

17.11.10 Layers Explorer

Displays the **Drawing Explorer** dialog box with the **Layers** category selected.

17.11.11 Layer States Explorer

Displays the **Drawing Explorer** dialog box with the **Layer States** category selected.

17.11.12 Hide xref symbols on/off

Allows you to toggle On/Off the display of xref layers.

17.11.13 Menu

Hide xref layers
When on, hides layers that came from externally referenced drawings.

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.

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.

It is possible to change the row height of the table view, by switching between **Compact View**, **Default View** and **Comfortable View**.

17.12 LAYERSTATE command

Opens the **Drawing Explorer** dialog box with **Layer States** selected.

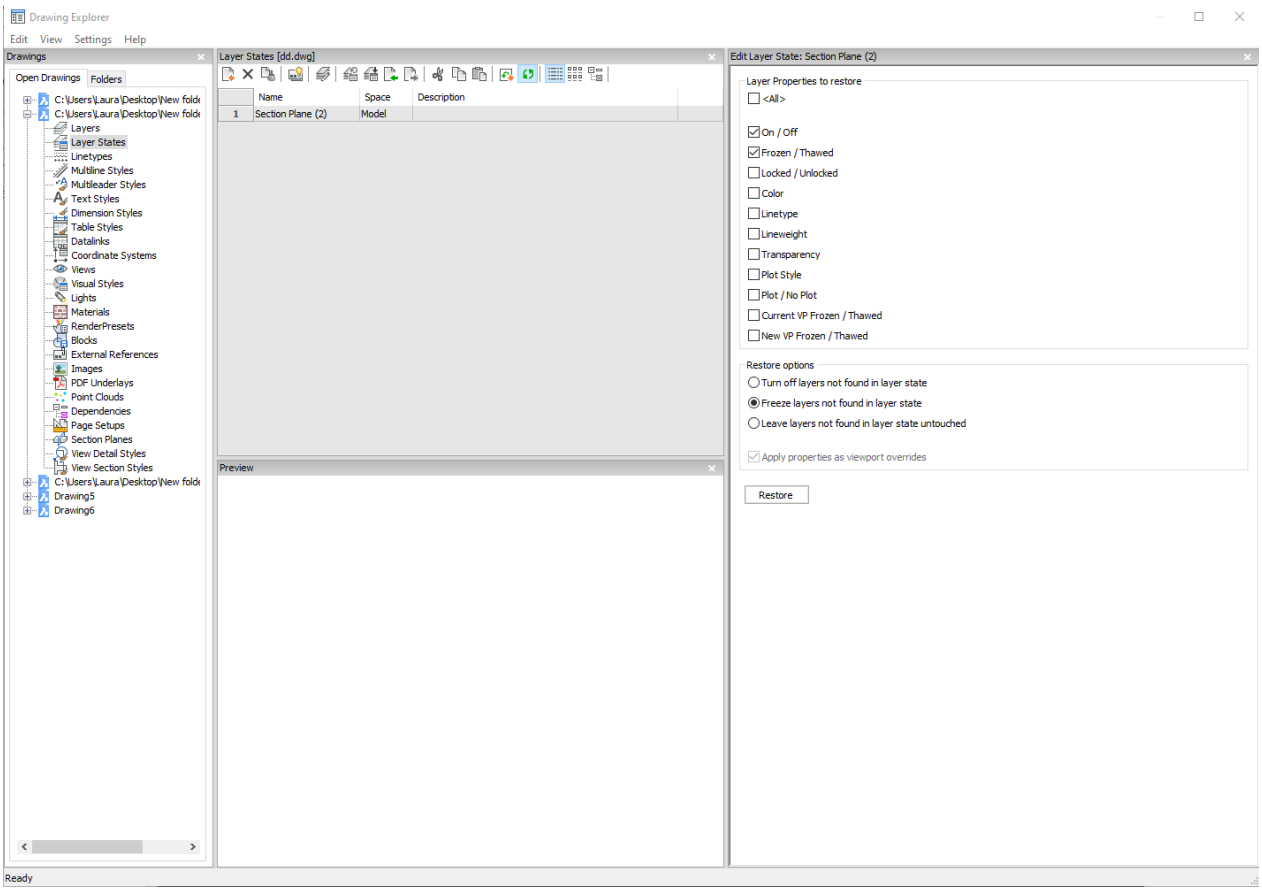
Shape Lite Pro Mechanical BIM

Icon: 

Alias: LAS

17.12.1 Description

Opens the **Drawing Explorer** dialog box on the Layer States category to manage layer states of layers that are used in the selected drawing.



17.12.2 Options within the command

<All>
Selects or deselects all properties. See the LAYER command for the meaning of the properties

Restore options
Determines what happens with layers not found in the layer state such as those added after the layer state was created.

Apply properties as viewport overrides
Applies the layer state as viewport overrides (see VP Layer properties) to the current layout viewport.

Note: This option is only available on a layout within a layout viewport.

17.12.3 Context Menu Options

New
Creates a new layers state with a generic name.

Delete
Deletes the selected layer state from the drawing.

Note: Layer states that are used in the drawing will be deleted without warning.

Hide xref symbols
Toggles the visibility of xref symbols on and off.

Edit Layer State
Displays the **Edit Layer State** dialog box to change the status of the layer state.

Add
Adds layers to layer state

Remove
Remove layers from layer state.

Restore
Restores the selected layer state to the default state.

Note: The properties of the layer will change after closing the **Drawing Explorer** dialog box.

Overwrite
Overrides the settings of the selected layer state with new settings changed by the **Edit Layer State** panel.

Import
Imports layer states from a LAS file through the **Layer States** dialog box.

Export
Exports layer states to a LAS file through the **Layer States** dialog box.

Note: The layer state file can be imported into other drawings or send to clients.

Rename
Renames the item.

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

17.13 LAYFRZ command

Freezes the layers of selected entities.

Shape Lite Pro Mechanical BIM

Icon:

17.13.1 Description

Freezes the layers of selected entities to hide all entities that are on the same layers as the selected entities.

17.13.2 Method

Select one or more entities on the layers you want to freeze. You can continue selecting entities until you press Enter to end the command. The Command line indicates which layers have been frozen:

- **This layer has been frozen: LayerName1**
- **This layer has been frozen: LayerName2**

If any of the selected entities are on the current layer, the Command line indicates that layer can't be frozen:

Cannot freeze layer: LayerName3. It is the current layer.

17.13.3 Options within the command

Settings
Allows you to choose to modify viewports or block selection settings.

Viewports
Allows you to specify the behavior for layout viewports.

- **Vpfreeze**: freezes layers in the current viewport. This is the default option.
- **Freeze**: freezes layers in all viewports.

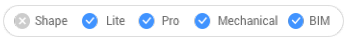
Block selection
Allows you to specify the behavior for blocks and external references.

- **Block**: freezes the layer of the selected block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is frozen. This is the default option.
- **Entity**: freezes the layer of the selected entity within a block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is frozen.
- **None**: this option behaves the same as the **Selection** option.
- **Selection**: freezes the layer of selected entities including blocks and external references. It ignores layers of entities within the block or external reference regardless of where you pick the block.

Undo
Undoes the previous LAYFRZ operation.

17.14 LAYISO command

Isolates the layers of selected entities.



Icon: 

17.14.1 Description

Isolates the layers of the selected entities, meaning only the layers of selected entities remain visible or unlocked, and all other layers will be hidden or locked.

17.14.2 Method

Select one or more entities on layers you want to isolate. You can continue selecting entities until you press Enter to end the command. If all selected entities are on the same layer, the Command line indicates which layer has been isolated and makes it the current layer:

This layer has been isolated: LayerName1. This layer is current: LayerName1.

If the selected entities are on more than one layer, the Command line indicates how many layers have been isolated.

Number of isolated layers: #.

If the selected entities don't include one from the current layer, the current layer is changed to one of the isolated layers.

17.14.3 Options within the command

Settings
Allows you to specify the behavior of layers that are not isolated.

Off
Hides entities on layers that are not isolated. You can specify how entities are hidden for layout viewports.

- **VPFreeze**: freezes layers in the current viewport.
- **Off**: freezes layers in all viewports. This is the default option.

Lock
Locks layers that are not isolated. This is the default option.

17.15 LAYLCK command

Locks the layer of a selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

17.15.1 Description

Lock the layer of a selected entity to prevent entities on that layer from being edited.

17.15.2 Method

Select an entity on the layer you want to lock.

The Command line indicates which layers have been locked:

- **This layer has been locked: LayerName1**
- **This layer has been locked: LayerName2**

Entities on locked layers are faded by default. You can change layer fading with the LAYLOCKFADECTL system variable.

17.16 LAYMCH command (Express Tools)

Changes the layer of selected entities to match a destination layer.

Shape Lite Pro Mechanical BIM

Icon: 

17.16.1 Method

Select entities to be changed and then select an entity on the target layer.

17.16.2 Options within the command

Type-it
Allows you to enter the layer name.

17.17 LAYMCUR command

Changes the working layer to that of the entity selected (short for "layer make current").

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SETLAYER

17.17.1 Method

Choose one entity whose layer will be made the current layer.

17.18 LAYMRG command (Express Tools)

Merges layers of selected entities into a destination layer.

Shape Lite Pro Mechanical BIM

Icon: 

17.18.1 Method

Select the entities on the layers to be merged and then select an entity on the target layer.

Note:

- The entities on the merged layers are moved to the target layer.
- The merged layers are purged from the drawing.

17.18.2 Options within the command

Type-it
Allows you to enter the layer name.

List
Allows you to enter the layer name(s) to list.

*****
Lists all available layers.

Do you want to continue?

Yes
The merged layers are purged.

No
Exits the command, without merging the layers.

17.19 LAYOFF command

Turns off the layers of selected entities.

Shape Lite Pro Mechanical BIM

Icon:

17.19.1 Description

Turns off the layers of selected entities to hide all entities that are on the same layers as the selected entities.

17.19.2 Method

Select one or more entities on the layers you want to turn off.

You can continue selecting entities until you press Enter to end the command. The Command line indicates which layers have been turned off:

- **This layer has been turned off: LayerName1**
- **This layer has been turned off: LayerName2**

If any of the selected entities are on the current layer, you are prompted:

This layer is current: LayerName3. Do you want to turn off current layer?

Specify if you want to turn the current layer off.

- **Yes**: turn off the current layer.
- **No**: do not turn off the current layer.

17.19.3 Options within the command

Settings

Allows you to choose to modify viewport or block selection settings.

Viewports

Allows you to specify the behavior for layout viewports.

- **Vpfreeze**: freezes layers in the current viewport. This is the default option.
- **Off**: turns off layers in all viewports.

Block selection

Allows you to specify the behavior for blocks and external references.

- **Block**: turns off the layer of the selected block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is turned off. This is the default option.
- **Entity**: turns off the layer of the selected entity within a block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is turned off.
- **None**: this option behaves the same as the **Selection** option.

- **Selection**: turns off the layer of selected entities including blocks and external references. It ignores layers of entities within the block or external reference regardless of where you pick the block.

Undo
Undoes the previous LAYOFF operation.

17.20 LAYON command

Turns on all layers in the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

17.20.1 Description

Turns on all layers in the drawing to view and edit entities on those layers.

The Command line indicates: **All layers have been turned on.**

Note: Entities on frozen layers are only visible if you also thaw the layer. Entities on locked layers are only editable if you also unlock the layer.

17.21 LAYOUT command

Creates, copies, renames, and deletes layouts.

Shape Lite Pro Mechanical BIM

Icon: 

17.21.1 Method

You can create an unlimited number of layouts in a single drawing. Each layout represents a sheet of paper.

17.21.2 Options within the command

?
Lists the names of layouts already defined in the drawing.

Set
Switches to the specified layout.

New
Creates new layouts.

Copy
Creates a new layout by making a copy of an existing layout.

Delete
Erases layouts from the drawing.

Rename
Renames layouts.

SAve
Saves the mentioned layout as DWG or DXF files format.

Template
Imports layouts from DWG, DWF, or DXF drawing files.
Specify the file name and the layout names.

neXt
Displays the next layout.

Previous
Displays the previous layout.

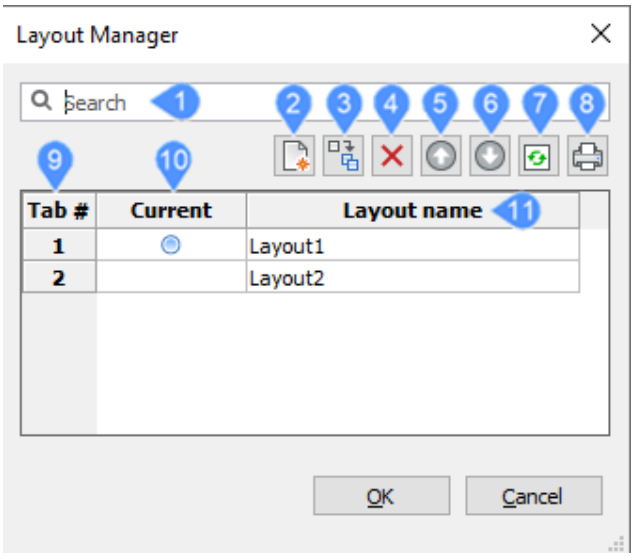
17.22 LAYOUTMANAGER command

Opens the **Layout manager** dialog box.

Shape Lite Pro Mechanical BIM

17.22.1 Description

Opens the **Layout manager** dialog box to view, create, copy and delete layouts in the current drawing.



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

17.22.2 Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

17.22.3 Add new layout

Creates a new layout.

17.22.4 Copy selected layout

Makes a copy of the selected layouts and adds them to the end of the list.

17.22.5 Delete

Deletes the selected layouts.

17.22.6 Move up

Moves the selected layouts up the list.

17.22.7 Move down

Moves the selected layouts down the list.

17.22.8 Clear selection

Removes highlighting from layout names, thereby unselecting them.

17.22.9 Publish

Opens the **Publish** dialog box.

17.22.10 Tab

Displays the number of the layout.

17.22.11 Current

Click in the **Current** column to make the layout current.

17.22.12 Layout name

Displays the name of the layout.

17.23 LAYOUTMERGE command (Express Tools)

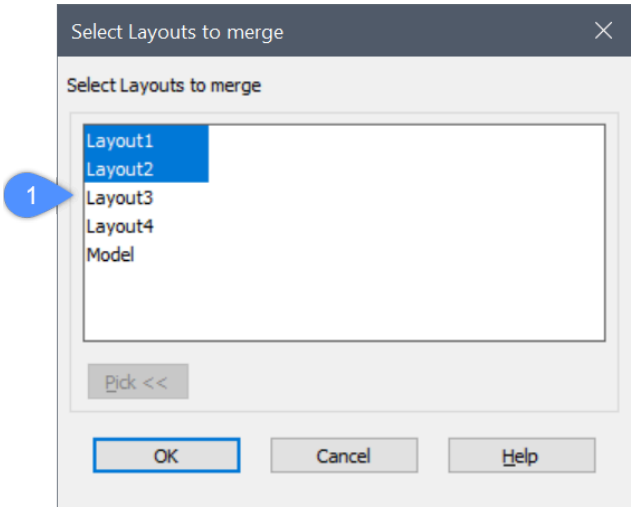
Merges entities from specified layouts into a destination layout.

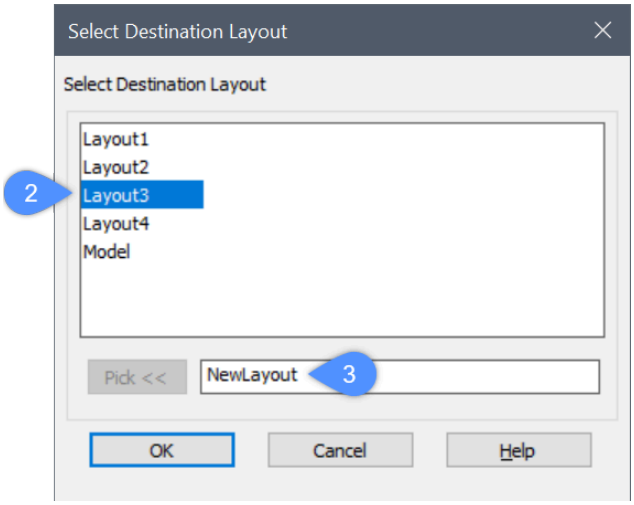
Shape Lite Pro Mechanical BIM

Icon: 

17.23.1 Method

Opens the **Select Layouts to merge** dialog box that allows you to merge entities from specified layouts into a destination layout, saving the corresponding views.





1 Layouts to merge

2 Destination layout

3 Type a layout name

17.23.2 Layouts to merge

Allows you to choose one or more layouts to be merged.

17.23.3 Destination layout

Allows you select a destination layout from the list.

17.23.4 Type a layout name

Allows you to type a layout name. If the layout does not exist, the **Create layout?** dialog box opens which asks you if you want to create the layout or not.

17.23.5 Options within the command

Remove empty layouts?
Allows you to choose to delete empty layouts or not.

17.24 -LAYOUTMERGE command (Express Tools)

Merges entities from specified layouts into a destination layout via Command line.

Shape Lite Pro Mechanical BIM

17.24.1 Options within the command

Select Layouts to merge
Allows you to specify the layout names to merge. You can type multiple layout names, separated by comma.

Select Destination Layout
Allows you to specify the name of the destination layout.

> ***Note:*** If the specified destination layout does not exist, the Command line prompt asks you if you want to create it.

Remove empty layouts?
Allows you to choose to delete empty layouts or not.

17.25 LAYTHW command

Thaws all layers in the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

17.25.1 Description

Thaws all layers in the drawing to view and edit entities on those layers.

The Command line indicates: **All layers have been thawed.**

> ***Note:*** Entities on layers are turned off are only visible if you also turn on the layer. Entities on locked layers are only editable if you also unlock the layer.

17.26 LAYTRANS command

Opens the **Layer translator** dialog box.

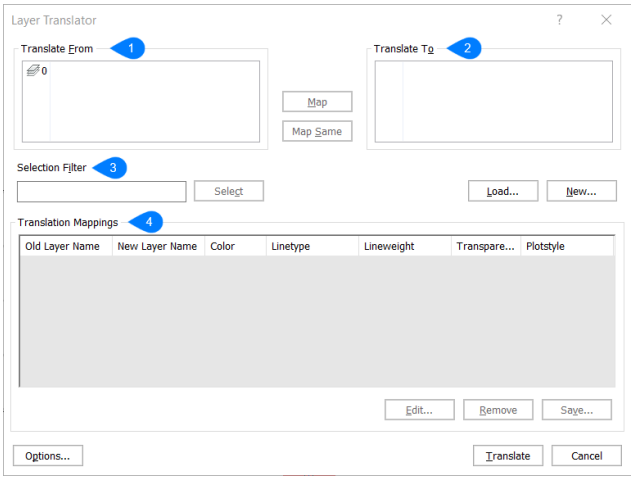
Shape Lite Pro Mechanical BIM

Icon: 

17.26.1 Description

Converts the layers in the current drawing to layers in other drawings.

Opens the **Layer translator** dialog box to translate layer properties by mapping to other layers.



1 Translate from

2 Translate to

3 Selection filter

4 Translation mappings

17.26.2 Translate from

Lists all the layer names in the current drawing.

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

Note: New layers can be created by simply entering the layer name and properties to which you want to map existing layer.

17.26.3 Translate to

Specifies which layer you want the chosen layer(s) to map to.

Note: 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.

17.26.4 Translation mappings

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

17.26.5 Options

Displays the **Options** dialog box, which offers additional controls for layer mapping.

17.27 LAYULK command

Unlocks the layer of a selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

17.27.1 Description

Unlocks the layer of a selected entity to enable entities on that layer to be edited.

17.27.2 Method

Select an entity on the layer you want to unlock. The Command line indicates which layers have been locked:

- **This layer has been locked: LayerName1**
- **This layer has been locked: LayerName2**

17.28 LAYUNISO command

Restores isolated layers to their previous state.

Shape Lite Pro Mechanical BIM

Icon: 

17.28.1 Description

Restores the Lock, On/Off, and VP Freeze properties of isolated layers as they were before you used the LAYISO command.

The Command line indicates: **Layers isolated by LAYISO command have been restored.**

17.29 LAYWALK command (Express Tools)

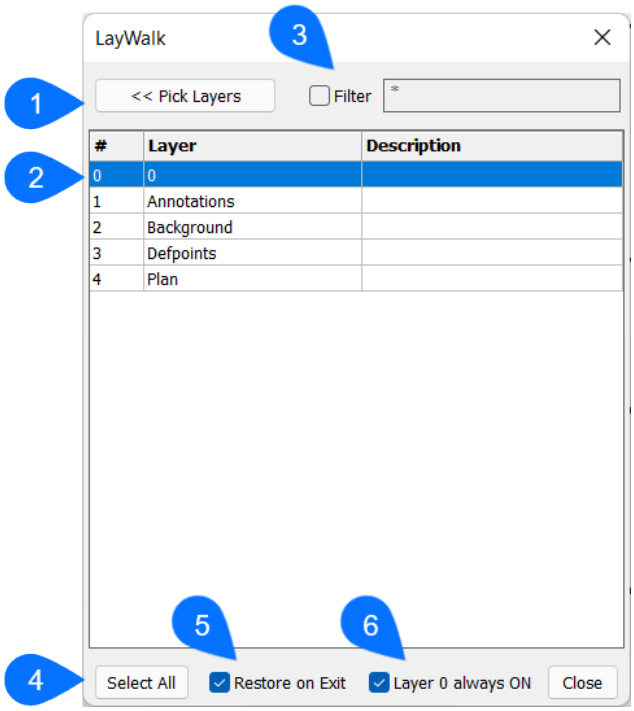
Displays selected layers and freezes all other layers.

Shape Lite Pro Mechanical BIM

Icon: 

17.29.1 Method

Opens the **LayWalk** dialog box which allows you to select the layers for which entities are displayed.



1 Pick Layers

2 Layer list

3 Filter

4 Select All

5 Restore on Exit

6 Layer 0 always ON

17.29.2 Pick Layers

Temporarily dismisses the **LayWalk** dialog box and allows you to select entities in the drawing, along with their layers.

17.29.3 Layer list

Displays a list of available layers. Select the layers for which you want to display the entities.

Note: The unselected layers are frozen.

17.29.4 Filter

Turns On and Off an active filter.

- Tick the checkbox to display only the layers that match the filter.
- Untick the checkbox to see the full list of layers.

17.29.5 Select All

Click this button to select and display all layers.

17.29.6 Restore on Exit

If the checkbox is ticked, it restores all layers to their previous state when closing the dialog box.

If the checkbox is unticked, the changes you made are saved.

17.29.7 Layer 0 always ON

If the checkbox is ticked, the layer 0 is always displayed.

If the checkbox is unticked, only the selected layers are displayed.

17.30 LCONNECT command

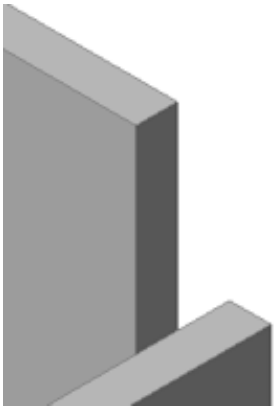
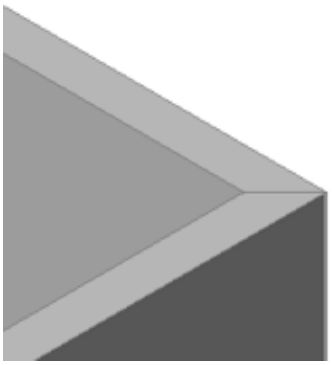
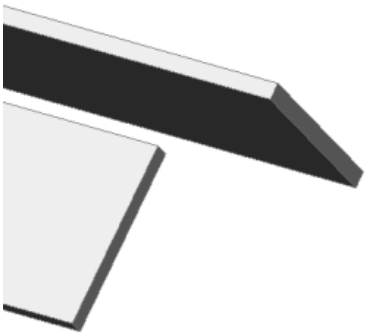
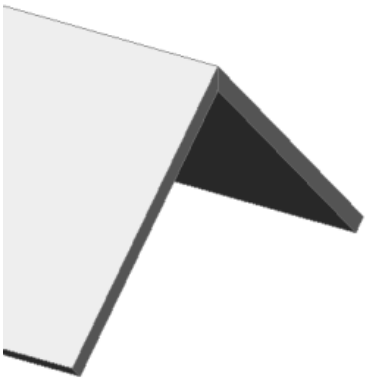
Creates or modifies an L-connection between solids.

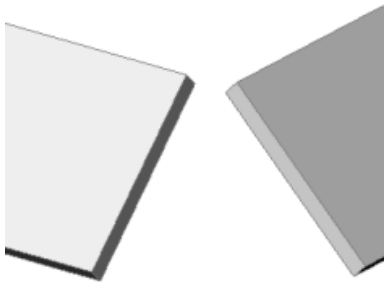
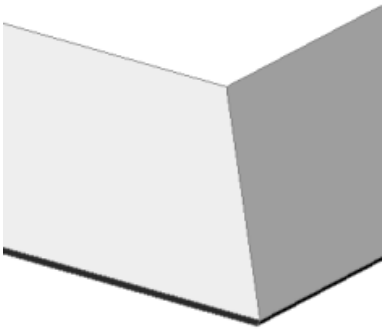
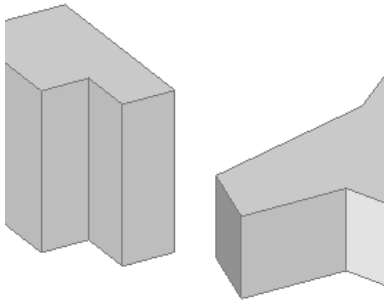
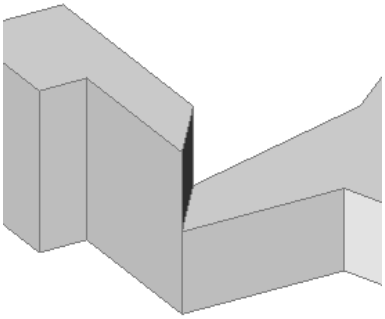
Shape Lite Pro Mechanical BIM

Icon:

17.30.1 Description

Creates or modifies an L-connection between solids. Optionally disconnects L-connected solids.

Before	After
	
	

Before	After
	
	

17.30.2 Options within the command

Select entities to connect
Manually select the two entities you want to connect.

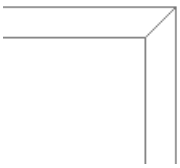
Switch
Allows to choose between two butted L-connections:



or



or a mitered L-connection:



or a touching (disconnected) connection:



Note: If the HOTKEYASSISTANT is set ON, press Ctrl key to cycle between the connection types.

17.31 LEADER command

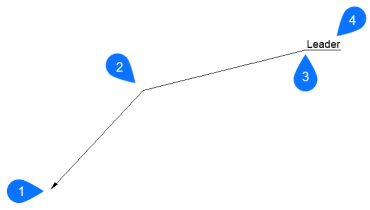
Draws leaders.

Shape Lite Pro Mechanical BIM

Alias: LE, LEAD

17.31.1 Description

Draws leaders by specifying several points.



1 Start point

2 Next point

3 To point

4 Annotation

Note: After you start creating a leader, options regarding the annotation may be available.

17.31.2 Options within the command

Format
Specifies the leader style.

Arrow
Draws an arrowhead at the leader's start point (default).

None
Does not draw an arrowhead.

SPline
Draws the leader as a spline.

STraight
Draws the leader as a straight-line segment (default).

Undo
Undoes the last leader line segment.

Annotation
Starts adding text to the end of the leader line.

Note: The annotation is created as an MTEXT entity.

Note: The annotation is independent of the leader line. When moving a leader, make sure to include the annotation in the selection set.

Block
Select a block in the drawing or upload a block file from the computer.

Copy
Select a Mtext, text, block reference or tolerance object from the drawing to use as annotation of the leader.

None
The leader command closes without annotation.

Tolerance
Enter the annotation as a tolerance notation through the **Geometric Tolerance** dialog box.

Mtext
Enter annotation as Mtext through a text formatting toolbar.

17.32 LEICACONVERT command

Converts or imports entities from drawings with Leica Infinity data to BricsCAD Civil objects.

Shape Lite Pro Mechanical BIM

Icon:

17.32.1 Description

Points and Texts are converted to Civil points. Meshes and Polyface Meshes are converted to TIN Surfaces.

You have to decide whether to delete the original entities or not.

17.32.2 Options within the command

Import
Imports Leica Infinity data entities (layers, linetypes, colors, and native CAD objects) from external drawings.

cOnvert
Converts Leica Infinity data entities from current drawings to BricsCAD Civil objects.

Yes
Deletes the points, texts, meshes, and polyface meshes entities from Leica Infinity data.

No
Keeps the points, texts, meshes, and polyface meshes entities from Leica Infinity data.

Cancel
Cancels the command.

17.33 LENGTHEN command

Changes the length of open objects such as lines, polyline segments and arcs.

Shape Lite Pro Mechanical BIM

Icon:

Alias: EDITLEN, LEN

17.33.1 Method

There are four methods to change an object's length.

- Dynamic
- Increment
- Percent
- Total length

Note: By selecting an object, the current length is listed in the Command line.

17.33.2 Options within the command

DYnamic
Specify a point to define the starting point of the object.

Note: The direction of the object won't change.

edit Mode
Returns to original prompt to switch lengthening modes.

Increment
Changes the length by a specified amount.

Angle
Changes the angle by a specified amount.

Percent
Changes the length of entities by a percentage.

Note: : For example, enter 25 (for 25%), and a 1"-long line is shorted to 0.25". Enter 150%, and the 1" line becomes 1.5".

Total
Enter the new total length of the object.

17.34 LIBRARYPANELCLOSE command

Closes the **Library** panel.

Shape Lite Pro Mechanical BIM

17.34.1 Description

Closes the **Library** panel to hide it from the current workspace. If the **Library** panel is stacked when you close it, the **Library** tab or icon is removed from the stack.

17.35 LIBRARYPANELOPEN command

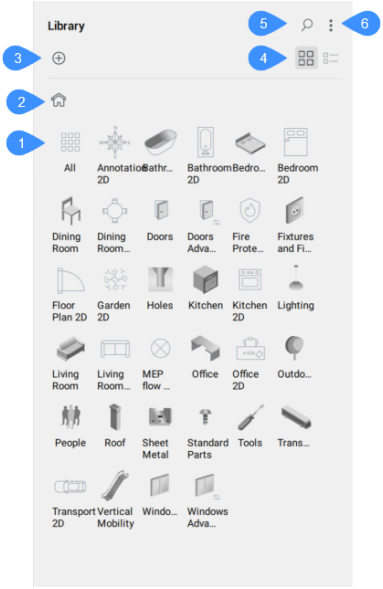
Opens the **Library** panel.

Shape Lite Pro Mechanical BIM

17.35.1 Description

Opens the **Library** panel to display it in the current workspace. The **Library** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Library** panel can be either floating, docked or stacked.

The **Library** panel offers a central location to access 2D and 3D block libraries.



1 Block library

2 Home

3 Add block to library

4 View

5 Search

6 Menu

17.35.2 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.

17.35.3 Home

Returns the block library to the home screen.

17.35.4 Add block to library

Opens the **Add block to library** dialog box.

Note: For information on how to use the **Library** panel, see the article **Using the library panel**.

17.35.5 View

Switches between grid and list views of the block content.

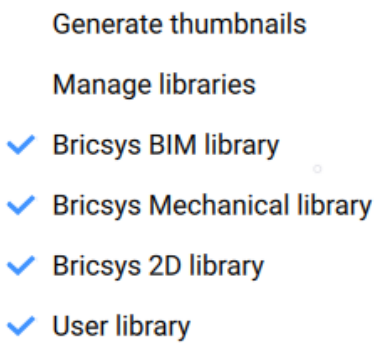
17.35.6 Search

Searches the library for words that you enter in the search box. Only displays the components that contain the entered sequence of characters.

Note: You can remove the active filter by pressing the close button on the right side of the search box.

17.35.7 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 ticked, the panel displays BIM components. The default path for BIM components is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\Bim\Components*.

Note: The parametric components under **Windows Advanced** and **Doors Advanced** folders allow for greater customization and contain hinge markers to be displayed on 2D sheets.

Bricsys Mechanical library

When ticked, the panel displays Mechanical components. The default path for Mechanical components is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\DesignLibrary*.

Bricsys 2D library

When ticked, the panel displays the content of the 2D library. The default path for the 2D library is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\Blocks2D*.

User library

When ticked, the panel displays the user defined components. The path for user library is defined by the COMPONENTSPATH system variable, which is by default *C:\ProgramData\Bricsys\Components*.

Edit the COMPONENTSPATH system variable in the **Settings** dialog box to add more paths. User library folders are added in the **Library** panel menu, where you can control their availability in the **Library** panel. When the path list is empty, the default path is added automatically.

17.36 LICENSEMANAGER command

Opens the **BricsCAD License Manager** dialog box.

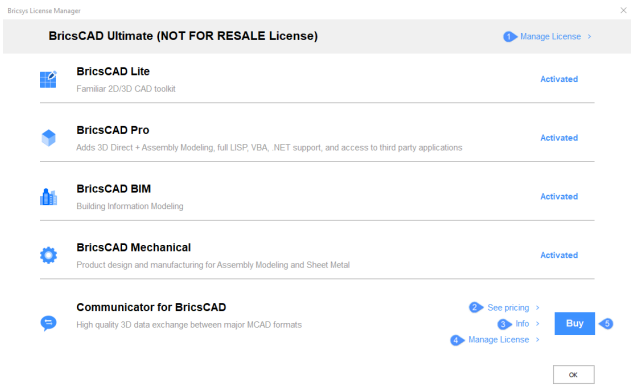
Shape Lite Pro Mechanical BIM

Icon:

17.36.1 Description

Opens the **Bricsys License Manager** dialog box to view and manage your BricsCAD license.

The **Bricsys License Manager** dialog box allows you to activate/deactivate the software licenses for BricsCAD and Communicator for BricsCAD®.



1 Manage License

2 See pricing

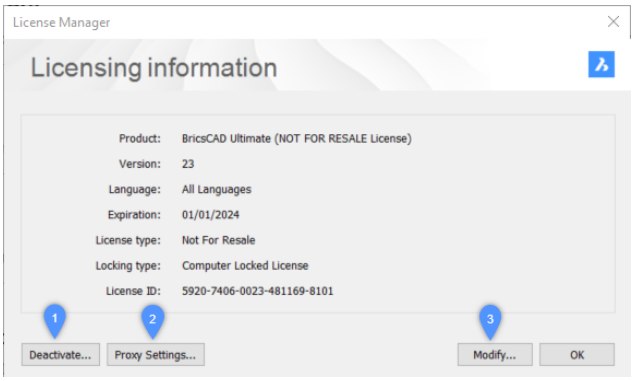
3 Info

4 Manage License for the selected product

5 Buy

17.36.2 Manage License

Opens the **License Manager** dialog box.



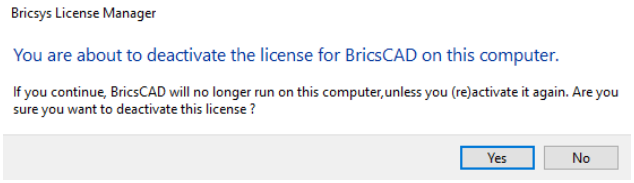
1 Deactivate

2 Proxy Settings…

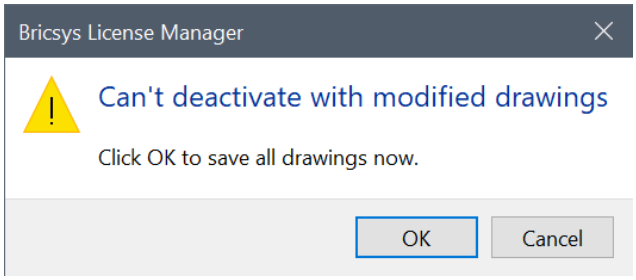
3 Modify…

Deactivate

Opens the **Bricsys License Manager** dialog box for confirming deactivation.



Note: If you have unsaved drawings, a warning message displays. You have to save all drawings before deactivating the license.



Important: The BricsCAD session closes after deactivating the license.

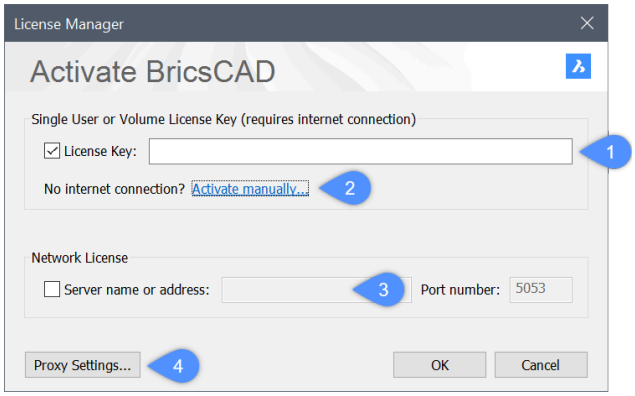
Proxy Settings…

Opens the **License Manager** dialog box where you can configure the proxy server.



Modify…

Opens the **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

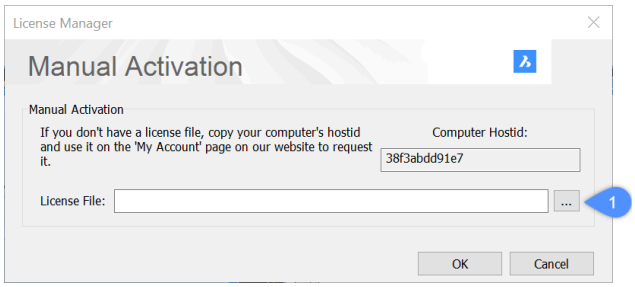
4 Proxy Settings…

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 **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\BricsCADV25.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.

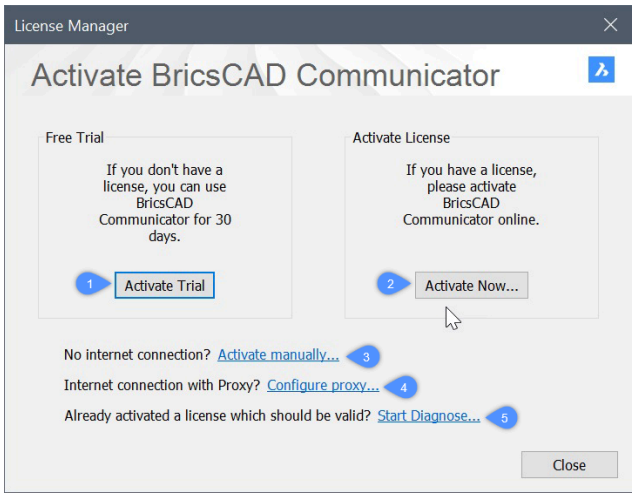
17.36.3 See pricing

Opens the Bricsys official website (Bricsys), where you can see the prices for Communicator for BricsCAD® and 1 year/3 years/lifetime BricsCAD subscription.

17.36.4 Info

Opens the **Communicator Info** dialog box. See **Communicator Info dialog box** article.

17.36.5 Manage License for the selected product



1 Activate Trial

2 Activate Now

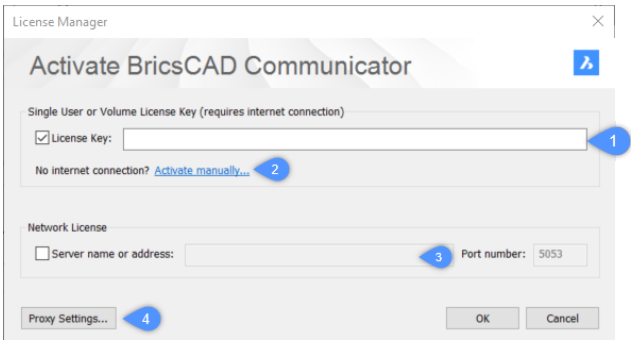
3 Activate manually…

4 Configure proxy…

5 Start Diagnose…

Activate Trial
Activates the 30 days free trial of Communicator for BricsCAD®.

Activate Now
Opens the **License Manager** dialog box for **Activate BricsCAD Communicator**.



1 License Key

2 Activate manually

3 Network License

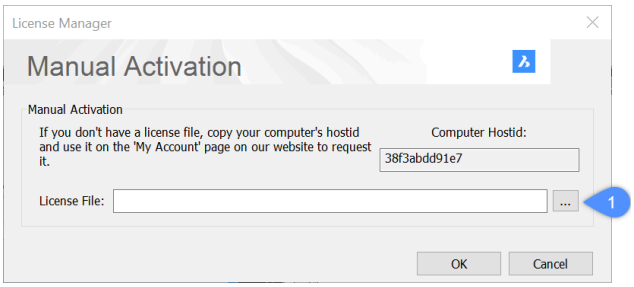
4 Proxy settings

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 **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\BricsCADV25.lic*



1 Select license file

Select license file

Opens the **Select license file** dialog box.

Network License

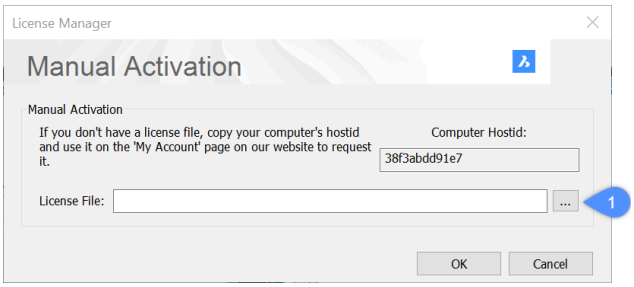
If you are using a network license, enter the host name or the IP address of the Network License Server.

Proxy Settings…

Opens the **License Manager** dialog box to configure the proxy server.

Activate manually…

Opens the **License Manager** dialog box for activating Communicator for BricsCAD®.



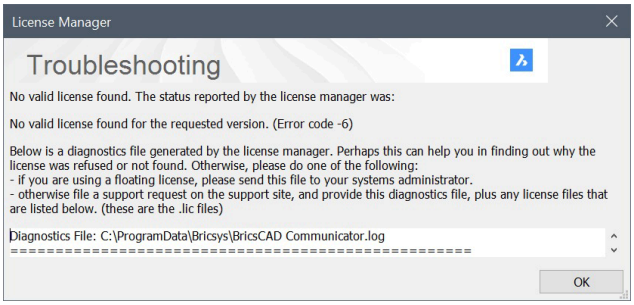
1 Select license file

Select license file
Opens the **Select license file** dialog box.

Configure proxy…
Opens the **License Manager** dialog box where you can configure the proxy server.



Start Diagnose…
Opens the **License Manager** dialog box for **Troubleshooting**.



17.36.6 Buy

Opens the Bricsys official website (Bricsys) where you can buy or subscribe to BricsCAD.

17.37 LIGHT command

Places light glyphs in drawings for generating more realistic renderings.

Shape Lite Pro Mechanical BIM

Icon:

Alias: LIGHTING

Note: Set DEFAULTLIGHTING system variable OFF to take into consideration the active lights sources defined in the drawing. Otherwise, only the default lighting will be used.

17.37.1 Options within the command

Point
Creates a point light.

Note: See the POINTLIGHT command for the meaning of its options.

Spot
Creates a spot light.

Note: See the SPOTLIGHT command for the meaning of its options.

Web
Creates a web light.

Note: See the WEBLIGHT command for the meaning of its options.

Distant
Creates a distant light.

Note: See DISTANTLIGHT command for the meaning of its options.

17.38 LIGHTLIST command

Opens the **Drawing Explorer** dialog box with **Lights** selected.

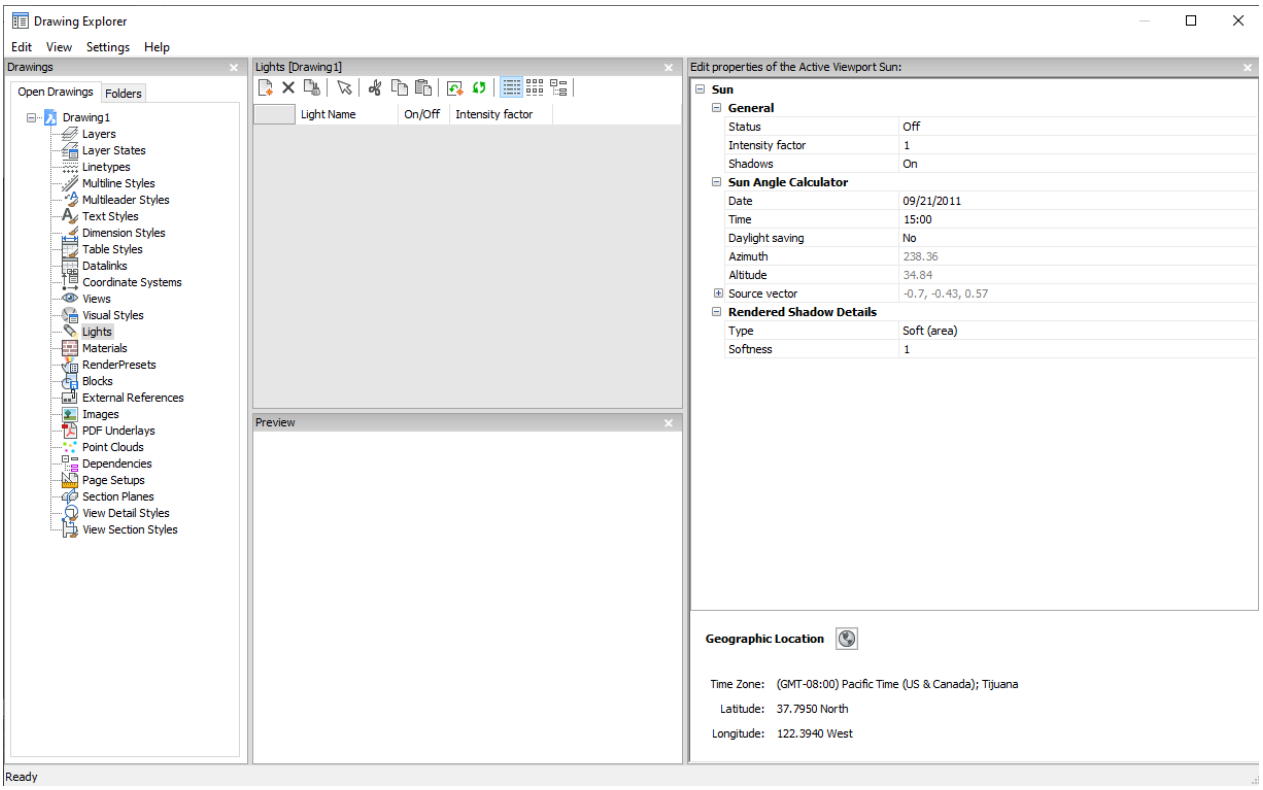
Shape Lite Pro Mechanical BIM

Icon: 

Alias: LL

17.38.1 Description

Opens the **Drawing Explorer** dialog box on the **Lights** category to manage the lights in the selected drawing.



17.38.2 Options within the Edit properties of the Active Viewport Sun panel

General
Defines general settings of the sun.

Intensity factor
Specifies an intensity factor that makes the light brighter or dimmer, at its source.

Sun Angle Calculator
Defines the sun angle by entering date, time and location information.

Rendered Shadow Details
Defines shadow rendering settings.

Geographic Location
Defines the graphic location.

Note: For more information about the properties of a distance light named Sun, which simulates sunlight, see SUNPROPERTIES command.

17.38.3 Context Menu Options

New
Creates a new light with a generic name.

Delete
Deletes the selected light from the drawing.

Note: Lights that are used in the drawing will be deleted without warning.

Select lights in the drawing
Selects a light through the model space of the drawing.

Rename
Renames the item.

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

17.39 LIMITS command

Sets an imaginary boundary for the extents of the drawing and optionally, of the grid.

Shape Lite Pro Mechanical BIM

Icon: 

17.39.1 Description

The command creates an imaginary rectangle boundary inside the drawing area.

Drawing outside of the area defined by this command is restricted when the LIMCHECK system variable is On. "Selected point is outside limits. Please select another point." will be reported in the Command line.

The imaginary boundary also limits the grid display when the first flag of the GRIDDISPLAY system variable is not set.

17.39.2 Options within the command

Lower left corner
Specifies the lower left corner of the limits.

Upper right corner
Specifies the upper right corner of the limits.

ON
Turn on the limits for your drawing area.

OFf
Turn off the limits for your drawing area.

17.40 LINE command

Creates line segments.

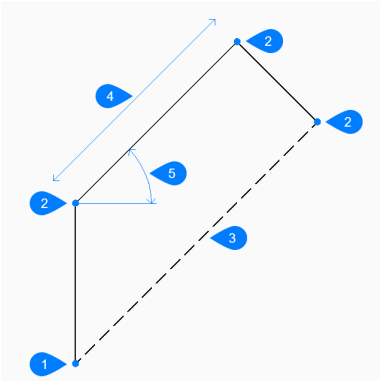
Shape Lite Pro Mechanical BIM

Icon: 

Alias: 3DLINE, L

17.40.1 Description

Creates a series of individual line entities by specifying the start and endpoint of each segment. Options allow you to specify the angle, undo, and close the geometry.



1 Start

2 End

3 Close

4 Length

5 Angle

17.40.2 Method

This command has 3 methods to begin creating a line segment:

- Start of line
- Last point
- Follow

17.40.3 Options within the command

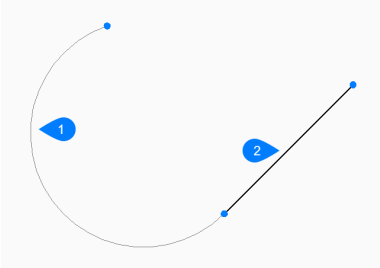
Start of line
Allows you to begin creating a line by specifying the start point.

Set end point
Specifies the end point of the line segment.

Note: You can continue adding unlimited segments until you press Enter to end the command.

Follow
Begins creating a line from the last drawn arc or line segment, following its angle.

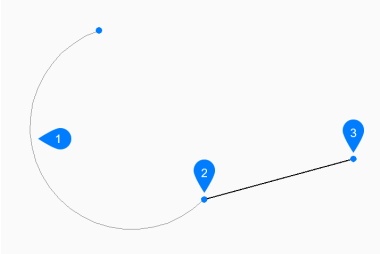
Length of line
Specifies the length of the line. Picking the endpoint only determines the length since the angle follows the previous segment.



1 Last drawn arc

2 Follow

Last point
Begins creating a line from the last point picked.



1 Last drawn arc

2 Last point

3 End point

Angle
Specifies the angle of the line segment.

Length
Specifies the length of the line segment.

Undo
Undoes the last line segment and continue drawing from its previous start point.

Close
Automatically draw a line from the end point of the last segment to the start point of the first segment.

17.41 -LINETYPE command

Loads, sets, and creates linetypes in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -LT

17.41.1 Method

You can create a new linetype by specifying a description (up to 47 characters long) and a linetype definition.

The linetype definition consists on a series of numbers separated by commas:

- Dashes: are shown by positive numbers.
- Spaces: are shown by negative numbers.
- Dots: are shown by number zero.

Note: After creating a new linetype, you need to load it to be accessible.

17.41.2 Options within the command

?
Displays the **Select Linetype File** dialog box, which allows you to select a LIN file.

Create
Displays the **Create or Append Linetype File** dialog box, which allows you to select a LIN file to add a new linetype.

Load
Displays the **Select Linetype File** dialog box, which allows you to load a linetype definition.

Set
Sets current a loaded linetype.

17.42 LINETYPE command

Opens the **Drawing Explorer** dialog box with **Linetypes** selected.

Shape Lite Pro Mechanical BIM

Icon:



Alias: DDLTYPE, EXPLTYPES, LT

17.42.1 Description

Opens the **Drawing Explorer** dialog box with the **Linetypes** category selected to view and modify linetypes in the current drawing.

New drawings contain at least these linetypes: Continuous, ByLayer, and ByBlock.

- **Continuous** linetype: displays entities with a solid, unbroken line.
- **ByLayer** linetype: displays entities with the linetype assigned to the current layer.
- **ByBlock** linetype: displays entities with the **Continuous** linetype until the entities are combined into a block definition.

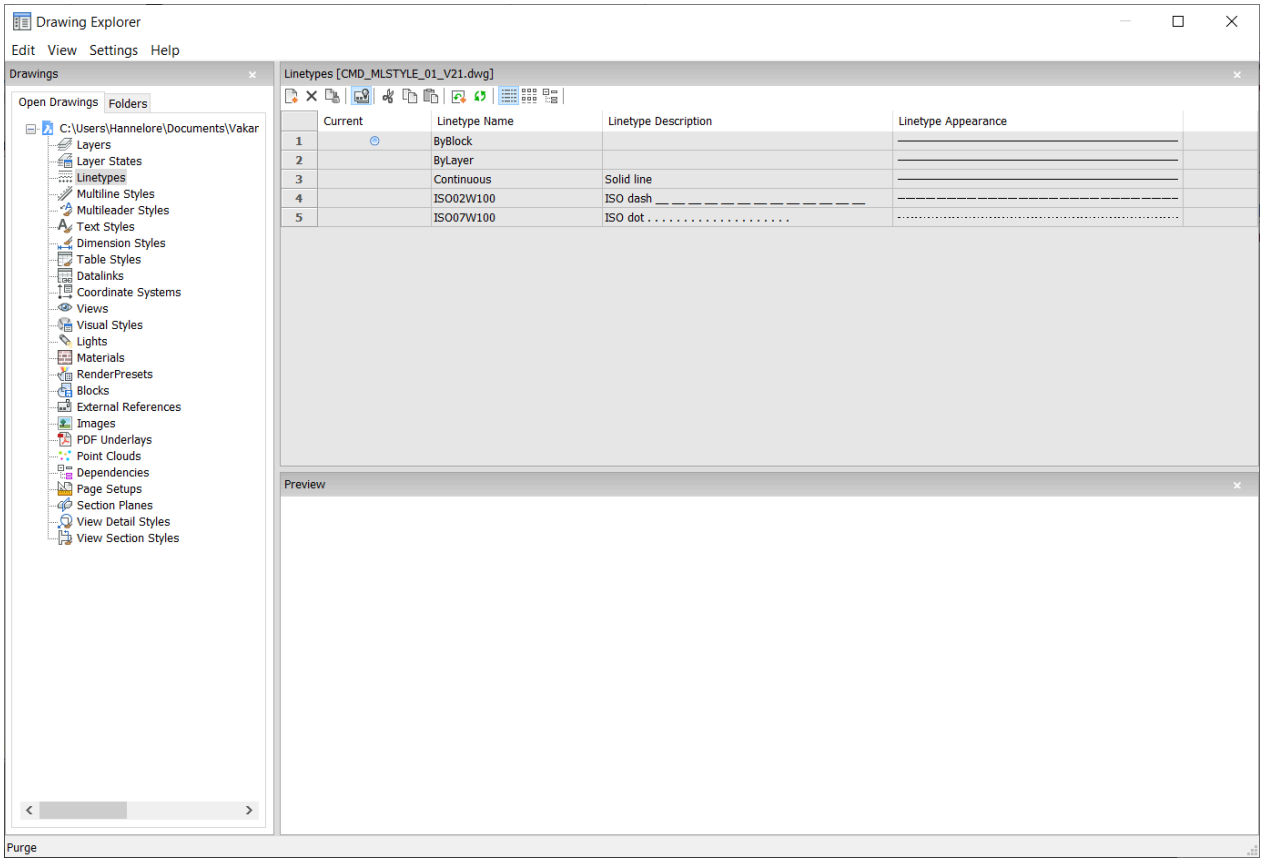
  Note:

  - If you need to control the linetype of a certain part of a block, you can assign the **ByBlock** value to that part of the block. This means that this part of the block does not have any value assigned until it is inserted into a drawing. You should assign the **ByBlock** value to entities before creating the block or change it in the block editor after.
  - When the block is inserted into the drawing, it displays the current linetype of the drawing for

those entities.

Any other linetype must be loaded into the drawing before it can be used: click the **New** button to load the linetype.

To start new drawings with all linetypes loaded, create and save the drawing as a DWT template file.



17.42.2 Context Menu Options

New

Loads additional linetypes into the drawing. Displays the **Load Linetypes** dialog box (see **Load Linetypes dialog box** article).

Delete

Deletes linetypes from the drawing.

Note: The following linetypes cannot be deleted:

- Continuous
- ByLayer
- ByBlock
- Any linetype in use

Rename

Renames the selected linetype.

Note: The following linetypes cannot be renamed:

- ByBlock
- ByLayer
- Continuous

Select all
Selects all linetypes.

Invert selection
Deselects the current selection and vice versa.

Set Current
Sets the selected linetype as the current one.

17.43 LIST command

Lists the properties of selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: LI, LS

17.43.1 Description

Returns a list of properties of the selected entities. A long lists of data could be generated, so press F2 to switch to the Prompt History window. When listing gets too long, press **Esc** key to end it.

Following properties are listed for each entity:

- Entity type
- Layer, Color, Linetype, lineweight: if these properties are not specifically defined, the information will be set to 'ByLayer'.
- Bounding box coordinates

17.43.2 Method

This command offers two methods to display the properties of a selected entity.

- Selecting entities before running the LIST command: Returns the properties of the selected entity.
- Running the LIST command before selecting entities: Allows to choose between different options before selecting the entities.

17.43.3 Options within the command

SOrt
Entities will be listed in sort of varying properties.

SEquential
Entities will be listed in the order by which they were selected.

Tracking
Specifies the number of lines of text to be displayed.

add to set (+)
Extra entities can be added, to list their properties.

subtract from set (-)
Allows to select the entity you want to remove from the selection.

Previous selection
Select the option in the Command line to go back to the previous selection.

select by PROperties
Lists the properties of entities with the same specified properties. Entities with the same property value for properties such as color, layer, linetype, name, thickness, type, value, width, handle, location, can be selected.

selection methoDs
Changes the selection method, same as the PICKAUTO system variable works.

17.44 LIVESECTION command

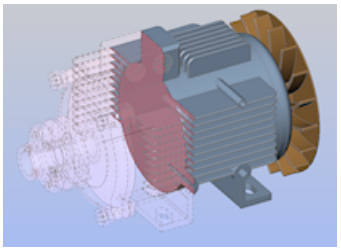
Toggles section views on/off.

Shape Lite Pro Mechanical BIM

Icon:

17.44.1 Description

Turns the display of the Live Section property of section planes on and off. When on, the insides of 3D models can be viewed.



Live sections differ from regular section planes by being interactive. This command requires at least one section plane in a drawing that was made with the SECTIONPLANE command.

Note: If the section was on, then it is turned off -- and vice versa.

Note: It is recommended to use the Clip Display property instead of Live Section. The Clip Display status can be set for multiple section entities simultaneously.

17.45 LMAN command (Express Tools)

Saves, edits and restores layer states.

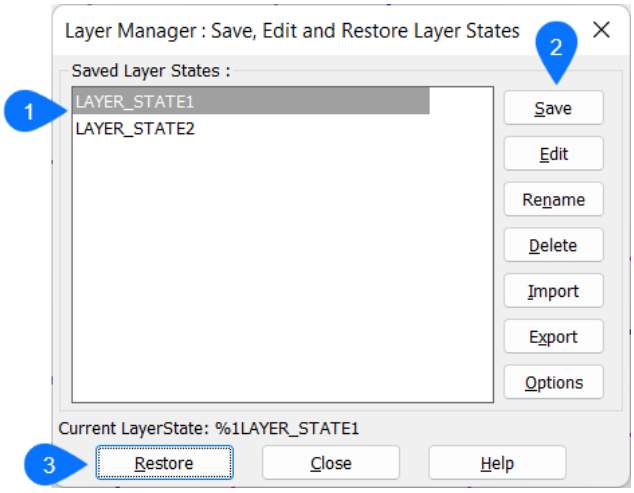
Shape Lite Pro Mechanical BIM

Icon:

17.45.1 Method

Opens the **Layer Manager: Save, Edit and Restore Layer States** dialog box which allows you to save, edit and restore layer states.

Layer states are stored in the drawing, but can also be exported or read from a .lay file.



1 Saved Layer States

2 Layer States options

3 Restore

17.45.2 Saved Layer States

Displays a list of saved layer states.

17.45.3 Layer States options

Save
Opens the **New LayerState Name** dialog box, which allows you to save the current layer status.

Edit
Allows you to edit the layers states by opening the **Drawing explorer** dialog box. The **New LayerState Name** dialog box opens, which allows you to save the changes.

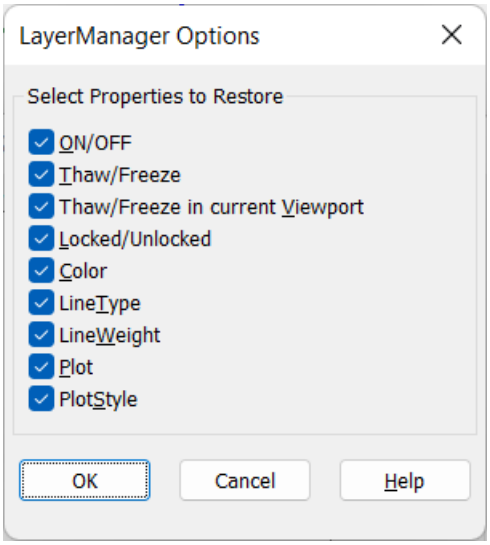
Rename
Opens the **Rename LayerState** dialog box, which allows you to rename a saved layer state.

Delete
Deletes the selected layer state. The **Warning** dialog box opens to confirm the action.

Import
Opens the **Import file name** dialog box, which allows you to choose a .lay file to open.

Export
Opens the **Export file name** dialog box, which allows you to export the selected layer state to a .lay file format.

Options
Opens the **LayerManager Options** dialog box, which allows you to set the options to be used for the **Restore** operation.



Note: All options are On by default.

17.45.4 Restore

Restores the layer settings of the selected layer state.

17.46 -LMAN command (Express Tools)

Saves, edits and restores layer states via the Command line.

Shape Lite Pro Mechanical BIM

17.46.1 Options within the command

?
Lists the available layer states.

Import
Opens the **Import file name** dialog box, which allows you to choose a LAY file to open.

Export
Opens the **Export file name** dialog box, which allows you to export the selected layer state to a LAY file format.

Save
Saves the current layer status.

Restore
Restores the layer settings of the mentioned layer state.

Delete
Deletes the mentioned layer state.

reName
Renames a saved layer status by specifying the old name and the new name.

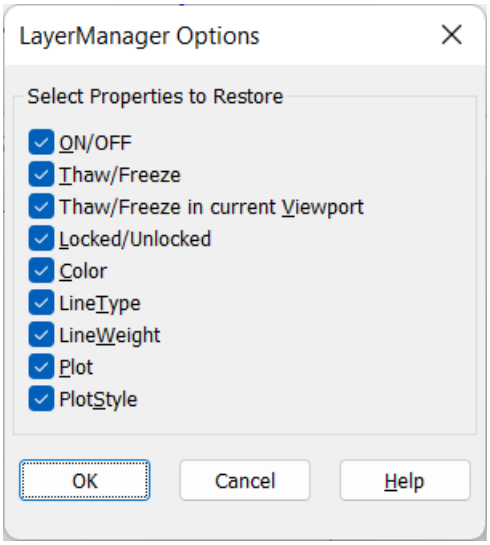
17.47 LMANMODE command (Express Tools)

Sets the layer manager options.

Shape Lite Pro Mechanical BIM

17.47.1 Method

Opens the **LayerManager Options** dialog box to select layer properties to restore.



17.48 -LMANMODE command (Express Tools)

Sets the layer manager options via the command line.

Shape Lite Pro Mechanical BIM

17.48.1 Description

Sets the LMAN mode setting by specifying a bit-code.

17.49 LOAD command

Opens the **Load shape file** dialog box.

Shape Lite Pro Mechanical BIM

17.49.1 Description

Opens the **Load shape file** dialog box to select an SHX file to load into the current drawing.

17.50 LOFT command

Passes through a series of cross sections to create 3D solids or surfaces.

Shape Lite Pro Mechanical BIM

Icon:

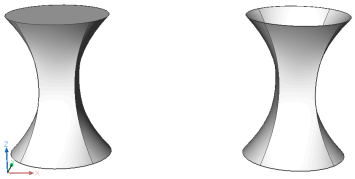
17.50.1 Method

Select the cross sections in lofting order. The shape of the resulting 3D solid or surface is defined by the cross sections.

Note: You have to specify at least two cross sections

17.50.2 Options within the command

Mode
Determines if the resulting loft is a solid or a surface.



Solid
Creates lofts as solid.

Note: Only for **Solid** mode, when the CREATESKETCHFEATURE system variable is ON, sketch features for loft are created in a dedicated BC_SKETCHES layer, which is not visible by default. The sketch features for loft (sketches, guide curves, and paths) are visible and editable as block references in the **Mechanical Browser** panel. Also, the loft features properties are present in the **Mechanical Browser** panel.

Note: The CREATESKETCHFEATURE system variable can also be controlled by pressing the **CreateSketchFeature** toggle button in the ribbon.

Surface
Creates lofts as surfaces.

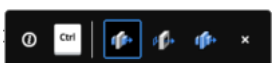
Create
Creates a loft entity. This is the default option.

Subtract
Subtracts the loft entity from the solids or surfaces that intersect with it.

Unite
Unites the loft entity with the solids or surfaces that intersect with it.

Note: The **Subtract** and **Unite** options are available only for solid loft entities created when the CREATESKETCHFEATURE system variable is ON.

Note: If the Hotkey Assistant (HKA) is ON, the Hotkey Assistant widget displays, indicating whether the command is in the **Create**, **Subtract**, or **Unite** mode. Repeatedly press the **Ctrl** key during the dynamic display of the loft to cycle the various options.



Guides
Uses both cross-sections and guide curves between the selected cross sections to create the loft.

Note: When DELOBJ system variable is set to 2, the selected guide entities are deleted.

Note: Invalid guide curves are ignored. If guide curves become invalid after editing, the geometry will be restored to its original state.

Path
Specifies the path curve.

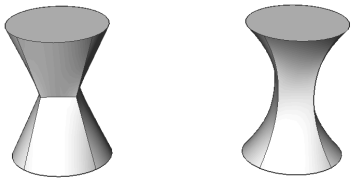
Cross sections only
Uses only cross section entities to create the loft, and no guides.

Settings
Sets variables that affect how the loft is constructed.

Ruled
Draws straight surfaces between the cross sections, features sharp edges at each cross-section.

Note: When the **Ruled** property is ON, the **Normal type** property becomes read-only.

Smooth fit
Draws smooth surfaces between the cross sections.



Normal to
Draws surfaces normal to the chosen cross sections. Select one of the options to become one of the following results:

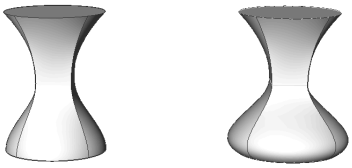
- Start
- Both
- End
- All

Note: LOFT entities created along a guide or path while the CREATESKETCHFEATURE system variable is ON have the **Normal type** property from the **Mechanical Browser** panel read-only.



Draft angles
Specifies the angles at the starting and ending cross-sections to modify the loft shape.

Note: The angle option specifies the angle at which the loft starts from a cross section. The magnitude option sets the relative distance of the surface from the cross section in the direction of the draft angle before surface starts to bend towards next section.



17.51 LOGFILEOFF command

Turns off log file recording.

Shape Lite Pro Mechanical BIM

17.51.1 Description

Log files record all prompts from the program and all of your keyboard input. It does not record mouse actions or other non-keyboard activities.

17.52 LOGFILEON command

Turns on log file recording.

Shape Lite Pro Mechanical BIM

17.52.1 Description

BricsCAD records all command text to the LOG file located in the folder specified by the LogFilePath system variable. The default name of the file is "drawingname_year-month-date_hour-minutes-seconds.log", such as drawing1_2029-08-31_08-32-46.log. Setting the LOGFILEMODE system variable to 1 has the same effect as the LOGFILEON command. You can open the resulting log file in Notepad or another text editor.

Note: Log files record all prompts from the program and all your keyboard input. It does not record mouse actions or other non-keyboard activities.

17.53 -LOGIN command

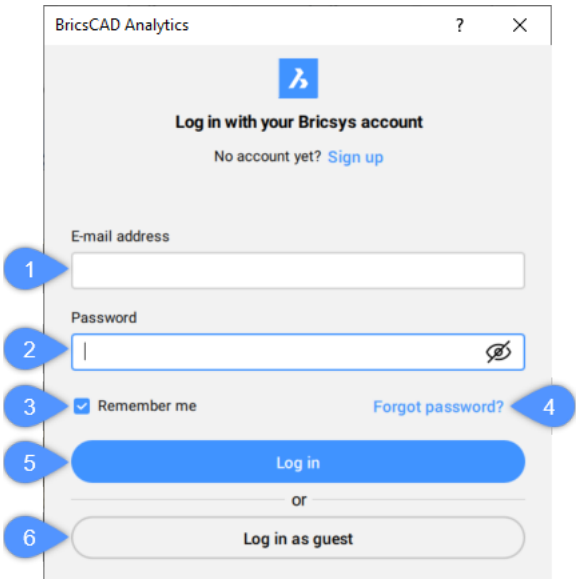
Authenticates to Bricsys server.

Shape Lite Pro Mechanical BIM

17.53.1 Description

Opens the **BricsCAD Analytics** dialog box to authenticate to Bricsys server.

The **BricsCAD Analytics** dialog box allows you to login to BricsCAD with your personal Bricsys account.



1 E-mail address

2 Password

3 Remember me

4 Forgot password?

5 Log in

6 Log in as guest

17.53.2 E-mail address

Specifies the email address connected to your Bricsys account.

17.53.3 Password

Specifies the password.

17.53.4 Remember me

Check this box to keep being logged in.

17.53.5 Forgot password?

Redirects you to an internet browser to set up a new password.

17.53.6 Log in

Logs you in with your personal Bricsys account.

17.53.7 Log in as guest

Allows you to log in as guest.

17.54 -LOGINUSAGEDATAPROGRAM command

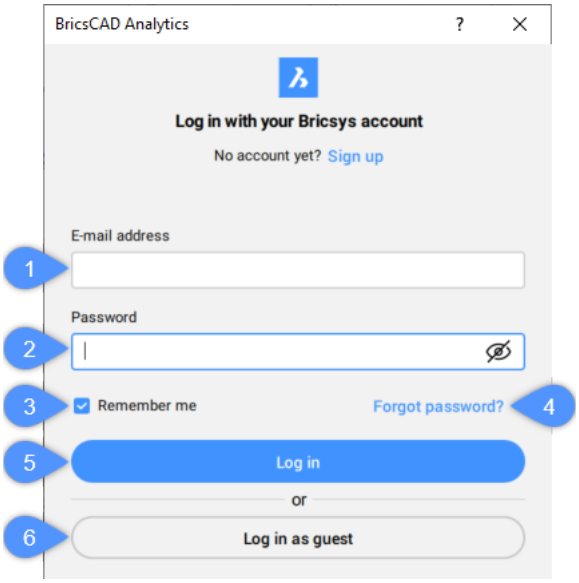
Opens the **BricsCAD Analytics** dialog box.

Shape Lite Pro Mechanical BIM

17.54.1 Description

Note: If you are already logged in, a login warning message displays. Press OK to continue.

The **BricsCAD Analytics** dialog box allows you to login to BricsCAD with your personal Bricsys account.



1 E-mail address

2 Password

3 Remember me

4 Forgot password?

5 Log in

6 Log in as guest

17.54.2 E-mail address

Specifies the email address connected to your Bricsys account.

17.54.3 Password

Specifies the password.

17.54.4 Remember me

Check this box to keep being logged in.

17.54.5 Forgot password?

Redirects you to an internet browser to set up a new password.

17.54.6 Log in

Logs you in with your personal Bricsys account.

17.54.7 Log in as guest

Allows you to log in as guest.

17.55 -LOGOUT command

Logs out of the Bricsys account.

Shape Lite Pro Mechanical BIM

17.55.1 Description

Logs the user out of the Bricsys account, which was used to access online services.

17.56 LOOKFROM command

Toggles LookFrom widget.

Shape Lite Pro Mechanical BIM

Alias: NAVVCUBE

17.56.1 Description

Changes the display of the LookFrom widget between on and off, and accesses the settings for displaying the widget.

17.56.2 Methods

There are two methods to control the widget:

- ON/OFf - toggle the widget's display.
- Settings - adjust the widget's look, by accessing the LookFrom Control section of the **Settings** dialog box.

17.57 LSP command (Express Tools)

Shows a list of all BricsCAD LISP commands, functions and variables.

Shape Lite Pro Mechanical BIM

17.57.1 Options within the command

Commands
Lists all BricsCAD LISP commands.

Functions
Lists all BricsCAD LISP functions.

Variables
Lists all BricsCAD LISP variables.

Load
Opens the **Load Application Files** dialog box to load and unload application files.

17.58 LSPSURF command (Express Tools)

Edits and debugs LISP applications.

Shape Lite Pro Mechanical BIM

17.58.1 Description

Opens the **BLADE - BricsCAD LISP Advanced Development Environment** dialog box to edit and debug LISP applications.

17.59 LWEIGHT command

Opens the **Settings** dialog box with the **Lineweights** category expanded.

Shape Lite Pro Mechanical BIM

17.59.1 Description

Opens the **Settings** dialog box with the **Lineweights** category expanded to view and modify relevant system variables.

18. M

18.1 MAIL command

Opens the default email client.

Shape Lite Pro Mechanical BIM

18.1.1 Description

Opens the default email client to automatically start a new email with the current drawing attached. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

18.2 MANAGEEXPERIMENTALFEATURES command

Allows you to enable/disable the **Experimental** mode.

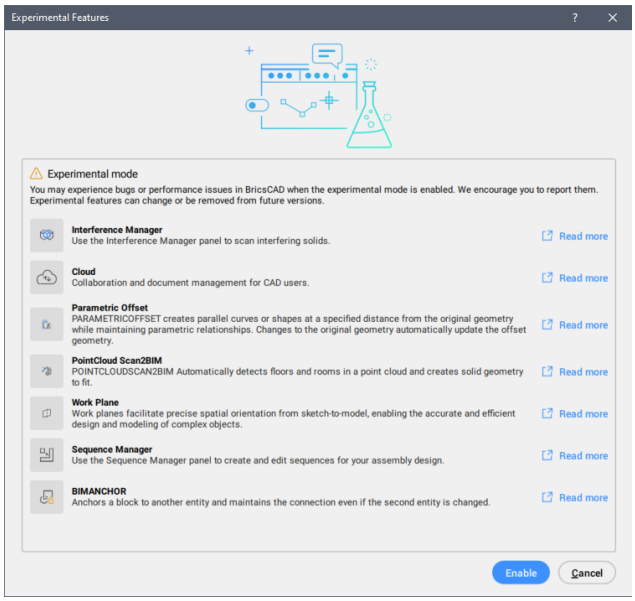
Shape Lite Pro Mechanical BIM

18.2.1 Description

Opens the **Experimental Features** dialog box to enable/disable the **Experimental mode**.

Note:

- The **Experimental mode** is disabled by default. Enabling the mode gives you access to early features, and provides you with the opportunity to help the BricsCAD team in defining the direction of these features.
- Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.



Note: Pressing the **Read more** button opens the relevant help page.

Important: You may experience bugs or performance issues in BricsCAD when the **Experimental mode** is enabled. We encourage you to report them. Experimental features can change or be removed from future versions.

The experimental features available at the moment are:

Interference Manager
Use the **Interference Manager** panel to scan interfering solids.

Cloud
Collaboration and document management for CAD users.

Parametric Offset
PARAMETRICOFFSET creates parallel curves or shapes at a specified distance from the original geometry while maintaining parametric relationships. Changes to the original geometry automatically update the offset geometry.

PointCloud Scan2BIM
Automatically detects floors and rooms in a point cloud and creates solid geometry to fit.

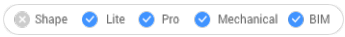
Work Plane
Work planes facilitate precise spatial orientation from sketch-to-model, enabling the accurate and efficient design and modeling of complex objects.

Sequence Manager
Use the **Sequence Manager** panel to create and edit sequences for your assembly design.

BIMANCHOR
Anchors a block to another entity and maintaining the connection even if the second entity is changed.

18.3 MANIPULATE command

Launches the manipulator widget to rotate, move, copy, mirror, and/or scale 2D entities and 3D models.



Icon: 

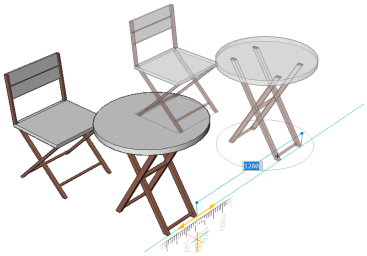
18.3.1 Methods

There are multiple methods to access the manipulator.

- Manipulate command.
- Select entities and press enter.
- Quad.
- Long press an object.
- Make sure that the MANIPULATOR system variable is set correctly.

18.3.2 Options within the command

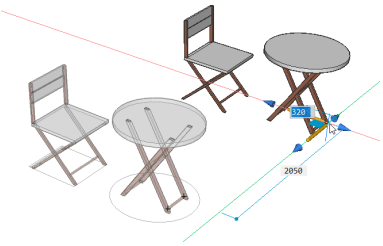
Select an axis
Moves the selection of entities along the selected axis.
Select an axis and enter the value of the displacement or define the new position by specifying a point.



Select a plane

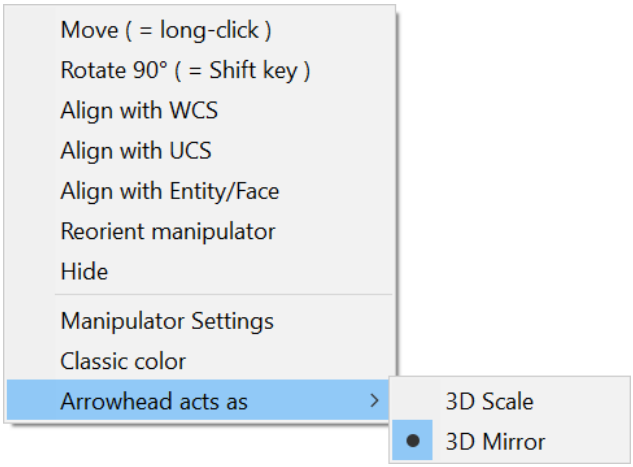
Moves the selection of entities in the selected plane.

Select a plane and enter the value of the displacement or define the new position by specifying a point.



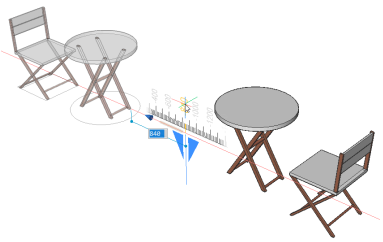
Select an arrowhead

Mirrors or scales the selection. By right-clicking when hovering over the manipulator widget, a contexts menu with other options displays. Here, the function of an arrowhead can be changed between 3D mirror and 3D scale.

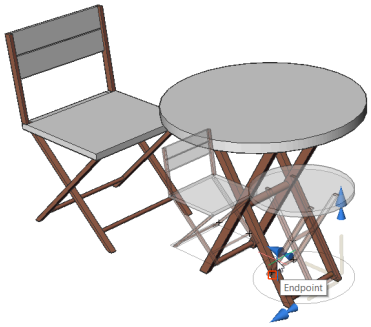


Select one of the arrowheads and specify the mirror axis or scale.

3D Mirror

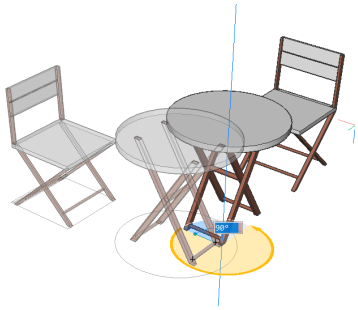


3D Scale



Select a rotation arc

Rotates the selection around one of the manipulator axes. Enter the rotation angle value or specify by clicking on a point in the drawing.



Select the anchor handle

- If MANIPULATORHANDLE = 0: Moves the manipulator. Specify a point to relocate.
- If MANIPULATORHANDLE = 1: Moves the selected entities unrestrictedly.
- Long-click the manipulator handle and move it to a new location.

Note: To copy the original entity so that a new entity is created, hold down the **Ctrl** key before starting the axis or make sure the copy option is enabled.

18.4 MAPCONNECT command

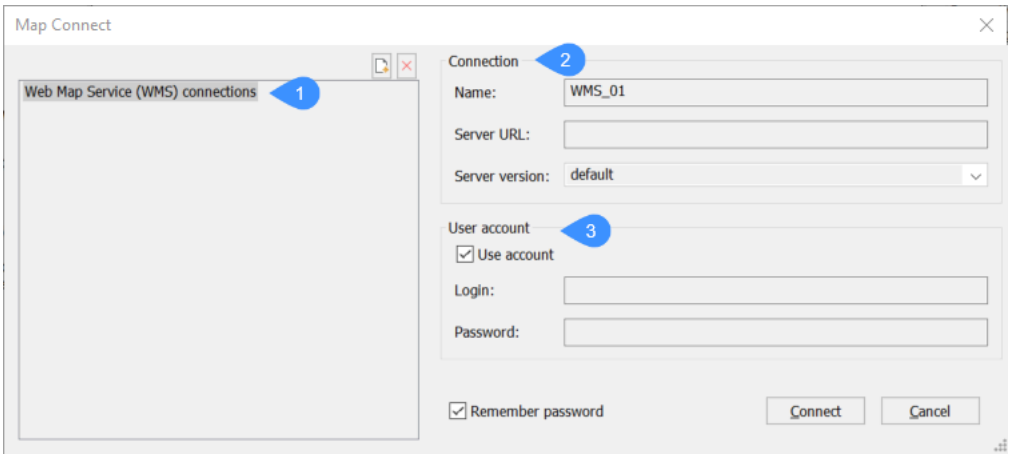
Allows to set up a connection with a Web Map Service.

Shape Lite Pro Mechanical BIM

18.4.1 Description

Displays the **Map Connect** dialog box to set up a connection with a Web Map Service.

Note: Before adding map data to a drawing, you must run the GEOGRAPHICLOCATION command to define the geographic location of the drawing.



1 WMS connections

2 Connection

3 User account

18.4.2 WMS connections

Displays a list of the different possible Web Map Service connections.

18.4.3 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.

18.4.4 User account

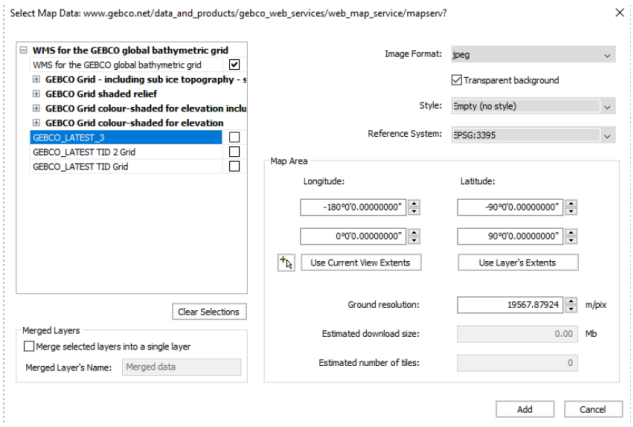
Allows you to log in to your account.

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.

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\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\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.

18.5 MAPTRIM command

Trims entities by a selected boundary.

Shape Lite Pro Mechanical BIM

Icon:

18.5.1 Description

Offers an easy and fast way to trim drawing entities, either inside or outside a specified boundary.

Note: The command enables to erase or ignore entities that cannot be trimmed, such as blocks and texts.

18.5.2 Method

There are two methods to define a boundary:

- Select an entity as a trim boundary.
- Define a polygon.

Entities accepted as trim boundary:

- Closed polylines
- Circles

- Closed splines
- Ellipses

Note: Entities that cannot be trimmed:

- Leaders
- Blocks
- Texts
- Mtexts
- 3d faces
- Solids
- Wipeouts
- Raster images
- Surfaces
- Regions
- Dimensions
- Hatches
- 3d solids
- Polyface meshes
- Polygon meshes

18.5.3 Options within the command

Select a trim boundary
Allows you to select an existing entity in the drawing as the boundary polygon.

Define
Allows you to define points as the boundary polygon.

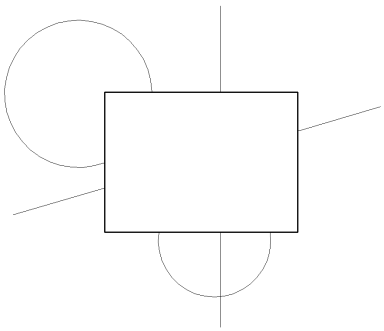
Undo
Undoes the last endpoint of line.

Select entities to trim
Allows you to selects entities to trim in the drawing.

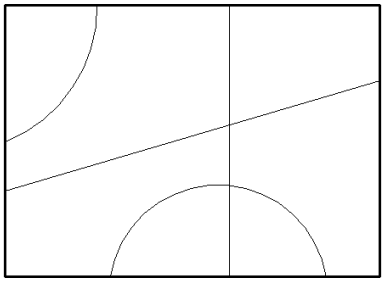
Automatic
Automatically selects all entities in the drawing to be trimmed.

Note: **Automatic** is the default option for entity selection.

Inside
Trims the entities inside the defined boundary.



Outside
Trims the entities outside the defined boundary.



Note: **Outside** is the default option to trim entities.

Ignore
Ignores the entities that cannot be trimmed.

Delete
Deletes the entities that cannot be trimmed.

Reference
If the **Inside** option was chosen, the entity that cannot be trimmed is deleted only if its insertion point lies inside the specified boundary.
If the **Outside** option was chosen, the entity that cannot be trimmed is deleted only if its insertion point lies outside the specified boundary.

18.6 MASSPROP command

Reports mathematical properties of 3D and 2D regions.

Shape Lite Pro Mechanical BIM

18.6.1 Description

Reports the area, perimeter and other mathematical properties of 3D solid models and 2D region entities.

Note: All other entities are ignored by the program.

Note: You can stop the calculation by pressing ESC.

18.6.2 Options within the command

Write analysis to a file?
Decides whether to save the report into a *.mpr file or not.

Note: If Yes, the **Create Mass** and **Area Properties File** dialog box opens. The *.mpr file can be opened in any text editor. MPR is short for Mass Properties Report.

18.7 MATBROWSERCLOSE command

Closes the **Render materials** panel.

Shape Lite Pro Mechanical BIM

18.7.1 Description

Closes the **Render materials** panel to hide it from the current workspace. If the **Render materials** panel is stacked when you close it, the **Render materials** tab or icon is removed from the stack.

18.8 MATBROWSEROPEN command

Opens the **Render materials** panel.

Shape Lite Pro Mechanical BIM

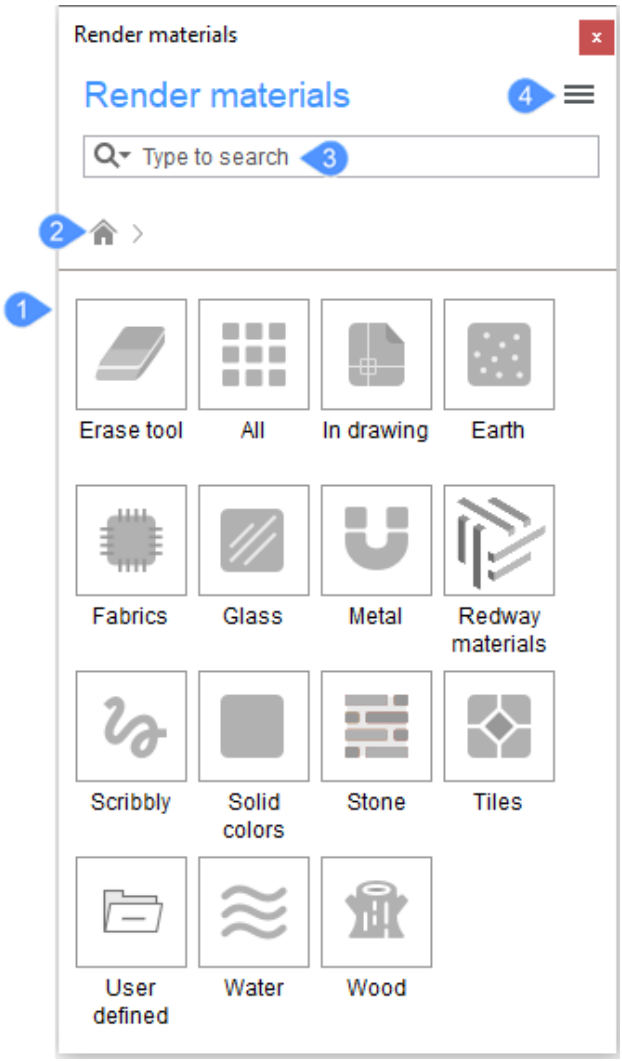
Alias: MATB

18.8.1 Description

Opens the **Render materials** panel to display it in the current workspace. The **Render materials** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Render materials** panel can be either floating, docked or stacked.

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 its properties.



1 Material library

2 Home

3 Search

4 Menu

18.8.2 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 on the screen. You can scroll through the materials using the mouse wheel or the scroll bar.

Note: Right-click a material to add it to library, to remove it from the drawing or to add it to the drawing.

18.8.3 Home

Returns the material library to the home screen.

18.8.4 Search

Searches the library for words you enter in the search box.

18.9 MATCHPERSPECTIVE command

Matches the current model space view with a background image.

Shape Lite Pro Mechanical BIM

18.9.1 Description

Matches the viewpoint of model space with the apparent perspective view provided by a background image, after three or more pairs of matching points are selected.

Before starting this command, place an image in the drawing with the BACKGROUND command, and then switch the viewpoint to perspective mode with the PERSPECTIVE variable.



18.10 MATCHPROP command

Copies selected properties and styles from one entity and applies it to one or more entities.

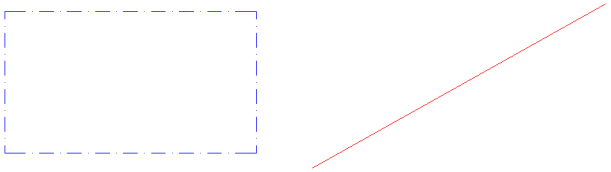
Shape Lite Pro Mechanical BIM

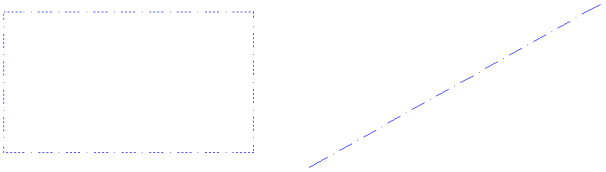
Icon:

Alias: MA

18.10.1 Method

Select the entity whose properties will be copied and the entities to apply the properties to.





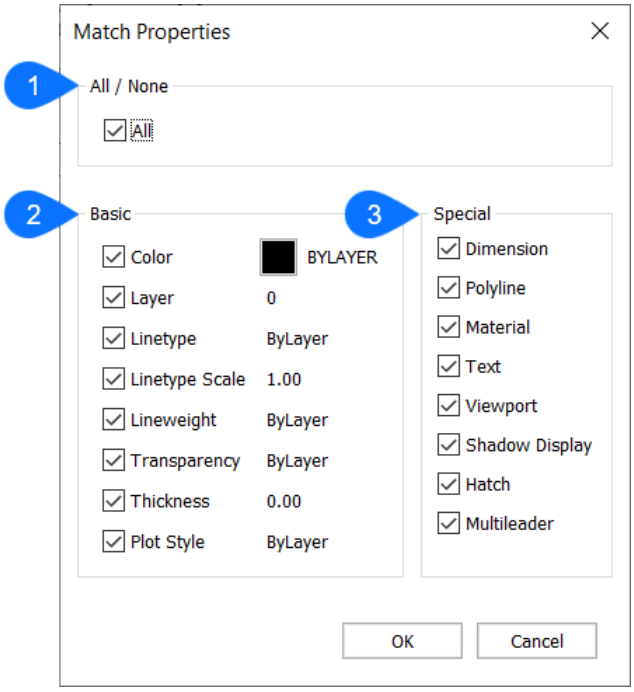
18.10.2 Options within the command

Settings

Specifies which settings to be copied. The **Match Properties** dialog box lists applicable properties along with their current value.

Check all properties that may be applied on other entities so not all properties are automatically matched.

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

18.10.3 All/None

When checked on, all properties are automatically selected. When checked off, all entities are deselected.

18.10.4 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.

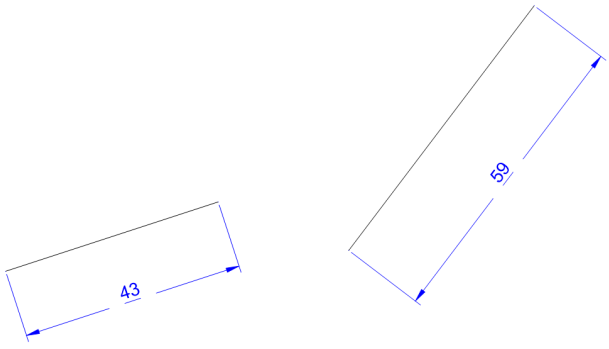
Thickness
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

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.

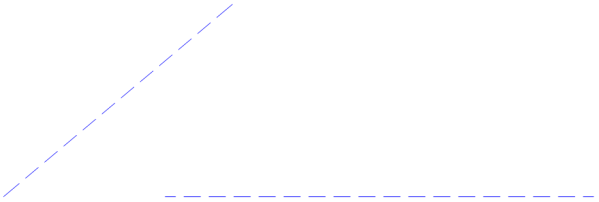
18.10.5 Special

Selects other less basic properties to copy.

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

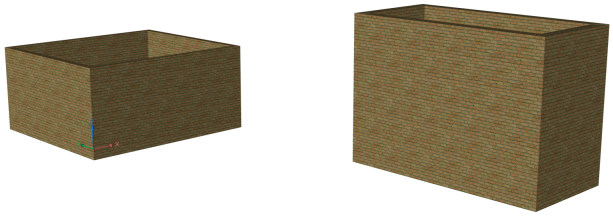


Polyline
When checked on, the width and linetype generation settings are copied. Other settings are ignored, such as taper and spline.



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.

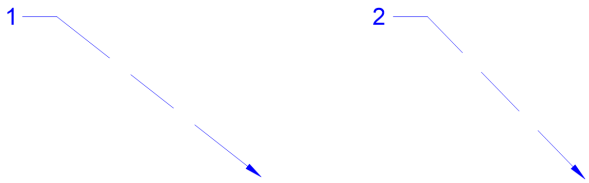
Hatch

When checked on, all hatch properties and annotative scaling are copied.



Multileader

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



18.11 MATERIALASSIGN command

Assigns current material to entities.

Shape Lite Pro Mechanical BIM

18.11.1 Description

Applies a material definition to filled entities. These materials make the results of visual styles and renderings look more realistic. They do not define physical properties, such as mass. Among visual styles, materials are displayed by the Modeling, Realistic, and Xray styles.

18.11.2 Method

Before a material can be assigned, one must be chosen. There are three methods of picking, assigning, and removing materials to and from entities.

Picking Materials

Follow one of these methods to specify the material to be applied:

- Specify a material with the CMATERIAL command:
- (Hidden option.) After starting the MATERIALASSIGN command, hold down the ALT key:

Use the dropper widget to select an entity that already has a material assigned to it.



Assigning Materials

Use the paintbrush glyph to apply the material to one or more valid entities.



Valid entities are 3D solids and 3D surfaces, as well as “solid filled” 2D entities, such as regions and traces. Materials cannot be assigned to unfilled entities, such as circles, nor to open entities, such as arcs, nor to areas that appear to be filled, such as those filled with hatches or gradients.

To apply the material to just one face instead of the entire entity, hold down the CTRL key.

Choose more entities, or undo the last material assignment or end the command.

Removing Materials

To remove materials, apply the **BYLAYER** material to entities or to layers.

18.12 MATERIALMAP command

Adjusts the placement of texture images.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SETUV

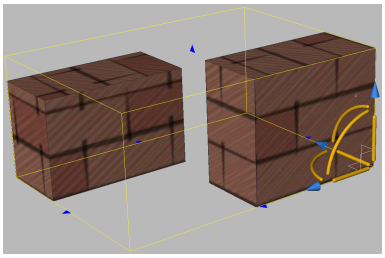
18.12.1 Description

Adjusts the placement of texture images onto 3D solids, 3D faces, wide polylines, and polygon meshes to create a more realistic look, such as matching a brick pattern to the wall of a house.

18.12.2 Method

The selected entities are included in a yellow bounding box. Blue grips and the manipulator widget appear:

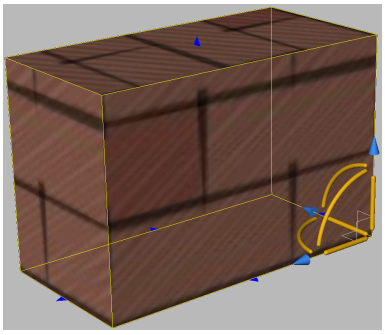
- Drag the blue grips to change the scale of the material horizontally and vertically.
- Click different parts of the manipulator to move, scale, and rotate the material.



18.12.3 Options within the command

Box

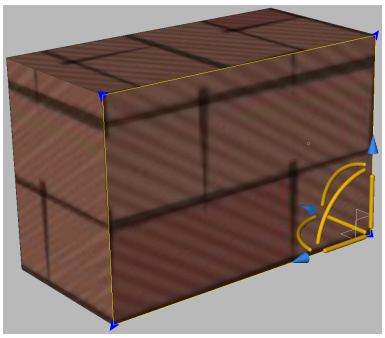
Wraps the texture around the six faces of the bounding box.



Planar

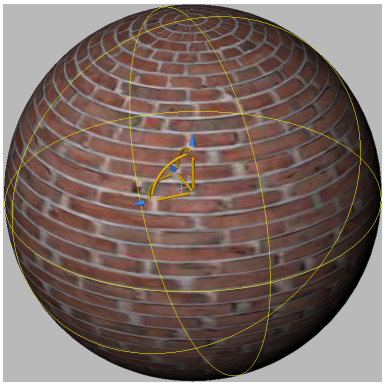
Aligns the texture to a single face.

Note: To select a face of a 3D solid, tick the Select Faces option of the SELECTIONMODES variable.



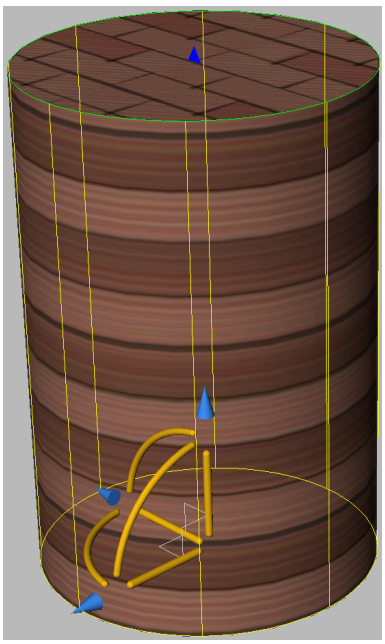
Spherical

Aligns the texture to a spherical shape. The top and bottom edges of the image are compressed to a point at the north and south poles of the sphere.



Cylindrical

Aligns the texture to a cylinder shape. The vertical edges of the image are wrapped together. The height of the image is scaled along the axis of the cylindrical shape.



sWitch mapping mode
Switches to a different mapping mode.

copY mapping to
Applies mapping from the source entity or face to the selected entities, thus duplicating the mapping, including any adjustments, to other entities. All entities share the same mapping origin, axes and scale. As a result, the texture image is extended seamlessly from one entity to another.

Reset mapping
Restores the default mapping.

18.13 MATERIALS command

Opens the **Drawing Explorer** dialog box with **Materials** selected.

Shape Lite Pro Mechanical BIM

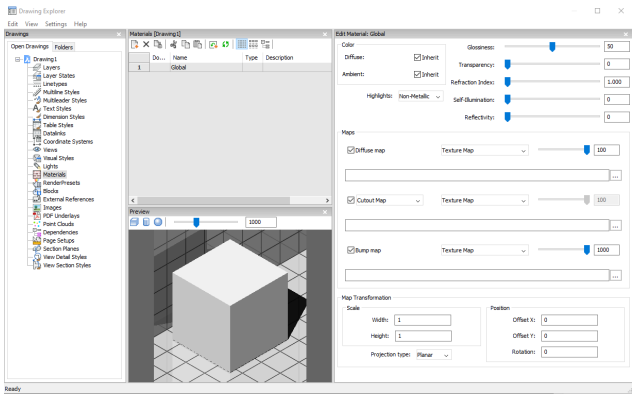
Icon:

Alias: FINISH, MAT, RMAT

18.13.1 Method

Open the **Drawing Explorer** dialog box with **Materials** category selected to view and modify materials in the current drawing.

Note: Materials provided by **RedWay** ( ) cannot be edited initially and so their settings are grayed-out, except for **Map Transformation**. To edit these materials, right-click a **Redway** material, and then choose **Convert to regular material** from the context menu.



18.13.2 Options within the Drawing Explorer

Preview options

Cube
Displays a preview of the selected material applied to a cube.

Cylinder
Displays a preview of the selected material applied to a cylinder.

Sphere
Displays a preview of the selected material applied to a sphere.

Light Intensity
Sets the light intensity in the Preview window.

Color

Diffuse
Sets the diffuse color. You can click the colored tile to choose a color in the **Color** dialog box. When **Inherit** is checked on, the entity color is applied.

Ambient
Sets the ambient color. You can click the colored tile to choose a color in the **Color** dialog box. When **Inherit** is checked on, the entity color is applied.

Highlights
Sets the highlight property of the material. You can choose between **Non-metallic** or **Metallic**.

Glossiness
Defines the glossiness of the material surface. You can choose a number in the range of 0 – 100.

Transparency
Defines the transparency of the material surface. You can choose a number in the range of 0 – 100.

Refraction Index
Defines the refraction index of the material surface. You can choose a number in the range of 1.00 – 3.00.

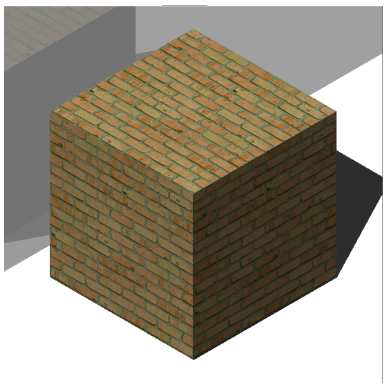
Self-Illumination
Defines the self-illumination of the material surface. You can choose a number in the range of 0 – 100.

Reflectivity
Defines the reflectivity of the material surface. You can choose a number in the range of 0 – 100.

Maps
Texture maps add detail to a surface, which is not included in the 3D model itself.

> ***Note:*** The **TextureMapPath** user preference defines the search path for texture map images. In the BricsCAD program folder there are 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 **Texture map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the **TextureMapPath** user preference to folder 1, 2 or 3.

Diffuse map
Diffuse maps apply a texture to the surface of a material such as wood grains, bricks, or tiles. You can select a texture bitmap file with the Browse button (⋮). The selected Texture Map can be applied as **Transparency Map** or as **Cutout Map**.



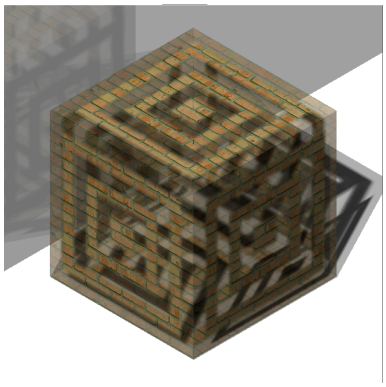
Blend factor

Sets the intensity of the texture map. In this way you can blend the texture map with the **Color** settings. You can set the **Blend factor** in a range of 0 to 100.

Transparency Map

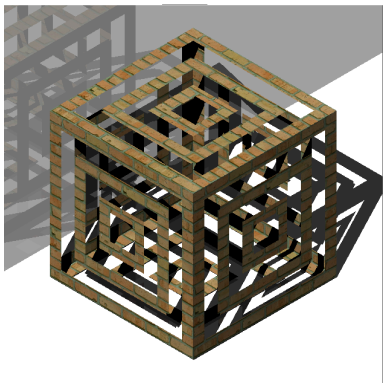
If **Transparency Map** is selected, the selected Texture Map image defines a transparency map. The image is recommended to be a grayscale image. White pixels will be invisible, black pixels will be opaque and gray pixels will be transparent.

Note: If you select a colored image, it will be converted to a grayscale image in the background. The alpha channel of the image is ignored.

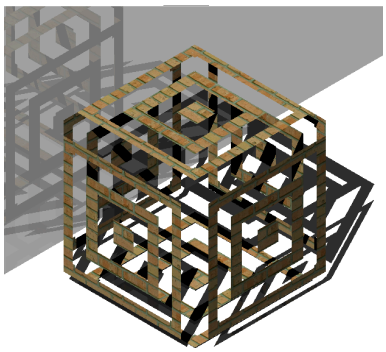


Cutout Map

If **Cutout Map** is selected, then the selected transparency map defines cutouts. White pixels are visible, and black pixels are invisible. The image is recommended to be a bitonal black-and-white image, without grays.



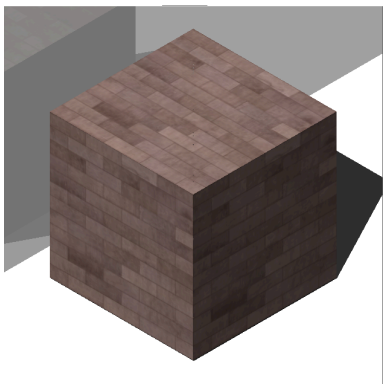
Cutout Map Inverted
Inverts the **Cutout Map** effect. Black pixels are visible and white pixels are invisible.

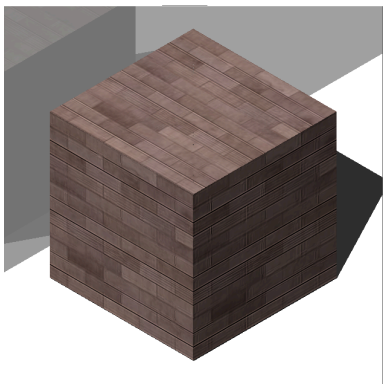


Note: If you select a colored or grayscale image, it will be converted to a black-and-white image in the background. The alpha channel of the image is ignored.

Blend factor
The transparency values in the texture will approach the scalar Transparency setting as the **Blend factor** approaches 0. This means that if the **Blend factor** is 100, then the transparency value will be controlled completely by the Texture Map image. If the **Blend factor** is 0, the transparency texture will be completely ignored and the scalar Transparency value will be used.

Bump map
Simulates bumps and wrinkles on the surface of an object. The result is a bumpy surface although the surface of the underlying object is not modified.





Left: Diffuse map only, Right: Bump Map applied

Blend factor
Defines the amount of bump which is applied to the material.

Map Transformation

Scale
The texture map image is applied at a size of 1 drawing unit multiplied by the **Width** and **Height** factors. For example, if the **Width** and **Height** factors or both set to 10, the size of the texture image is 10 x 10 drawing units.

Position

Offset
Texture maps are tiled starting from the origin of the WCS.
To adjust the tiling, you can define **Offset X** and **Offset Y** values. The offsets are expressed in drawing units.

Rotation
Sets the rotation of the texture maps.

Projection type
Defines the way the material is projected onto the object.

18.13.3 Context menu options

New
Creates a new material.

Delete
Deletes material definitions from the drawing. The following material definitions cannot be deleted:

- **Global**
- **Materials in use**

Rename
Renames the material.

Note: The following material cannot be renamed: **Global**.

Select All
Selects all material definitions.

Invert selection
Deselects the current selection and vice versa.

Convert to regular material
Converts materials to regular material and makes it possible to edit them.

Add material(s) to library
Adds materials to the material library so you can use them in other files as well.

18.14 MATLIB command

Opens the **Render materials** panel.

Shape Lite Pro Mechanical BIM

18.14.1 Description

Opens the **Render materials** panel to display it in the current workspace. The **Render materials** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Render materials** panel can be either floating, docked or stacked.

18.15 MEASURE command

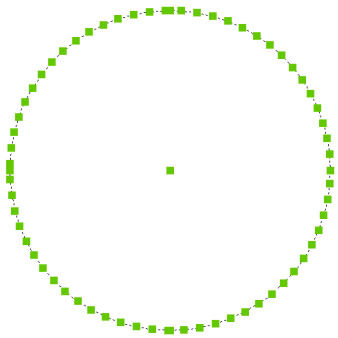
Places points or blocks along entities at equally-spaced specified distance.

Shape Lite Pro Mechanical BIM

Icon:

18.15.1 Method

Specify the entity you want to measure and enter a block distance value or insert a block. On the entity the points are places at equally-spaced specified distance.



Note: Points are usually invisible, so use the PDMODE system variable to make them larger.

18.15.2 Options within the command

Insert block
Inserts a block instead of a point.

Align blocks with entity?
Rotates the blocks to match the entity alignment.

18.16 MECHANICALBROWSERCLOSE command

Closes the **Mechanical Browser** panel.

Shape Lite Pro Mechanical BIM

18.16.1 Description

Closes the **Mechanical Browser** panel to hide it from the current workspace. If the **Mechanical Browser** panel is stacked when you close it, the **Mechanical Browser** tab or icon is removed from the stack.

18.17 MECHANICALBROWSEROPEN command

Opens the **Mechanical Browser** panel.

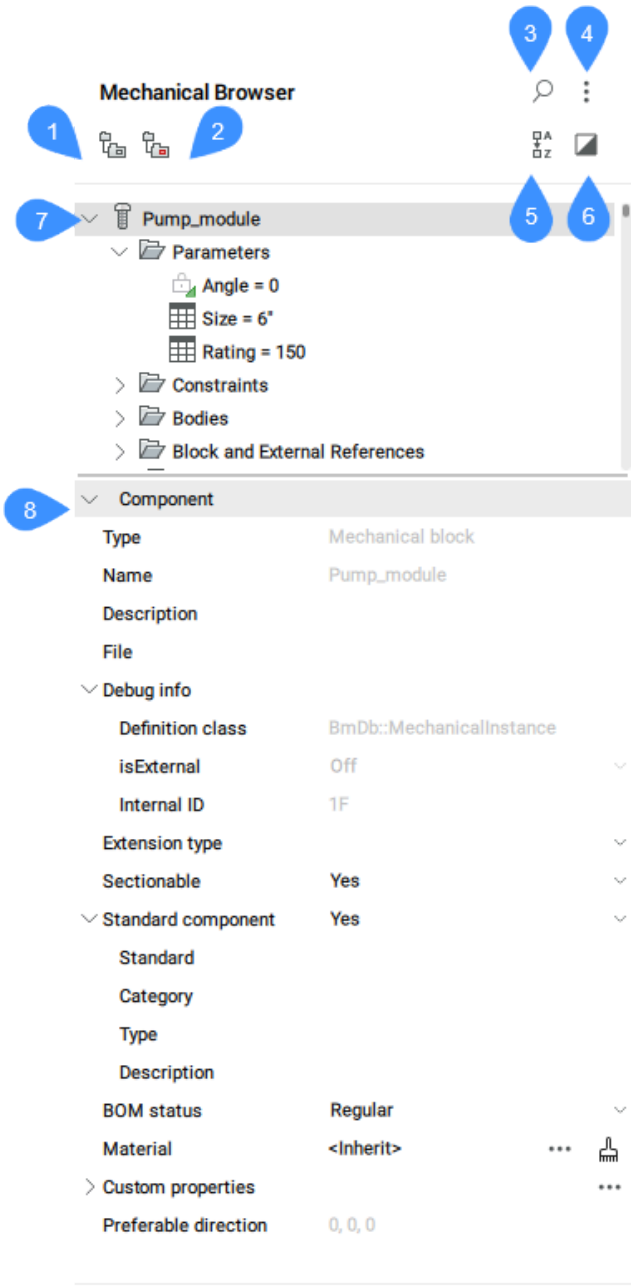
Shape Lite Pro Mechanical BIM

Icon: 

18.17.1 Description

Opens the **Mechanical Browser** panel to display it in the current workspace. The **Mechanical Browser** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Mechanical Browser** panel can be either floating, docked or stacked.

The **Mechanical Browser** panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Group by entity

2 Group by type

3 Show search

4 Settings

5 Sort Alphabetically

6 Show selected

7 Mechanical entities list

18.17.2 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

18.17.3 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

18.17.4 Show search

Opens the search field and the associated tools.

Search field
Searches the mechanical browser tree for the character string you enter. The search acts as a filter, so that the browser shows all components whose names or component names (or part of the names or component names) match the character string. Press **X** on the right of the search field to close the field.

18.17.5 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Controls the visibility of sub-component parameters.

Expressions of components parameters
Controls the visual representation of sub-component parameters: numeric value or assigned parameter name.

Sub-components of standard parts
Controls the visibility of sub-components of standard parts which are mechanical assemblies.

Arrays of entities
Toggles the display of Array nodes in the browser tree, including Components arrays.

Block and External references
Toggles the display of Blocks and External References nodes in the browser tree. Also, toggles node building in the **Bodies** folder for Block entities.

Always synchronize selection
Toggles the synchronization of the selection nodes in the **Mechanical Browser** with the selection of the corresponding objects in the model space. When an object is removed from a selection, only nodes corresponding to the objects remaining in the selection remain selected. If no objects are selected, the root node will be selected (similarly to when a document is open).

Keep values list order
Toggles the sorting of the values list for parameters inside the drop-down list.

Highlight selections
Toggles the highlighting of the selected entities in the drawing.

Asynchronous properties load
Controls if the **Mechanical Browser** collects properties asynchronously if more than one node is selected in the Mechanical Browser tree.

Expose all solids
Toggles the display of solids for mechanical blocks and solids not related to constraints under the **Bodies** folder.

Note: **Mechanical Browser** settings are controlled also by the MECHANICALBROWSERSETTINGS system variable.

18.17.6 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

18.17.7 Show selected

When enabled, only nodes corresponding to selected objects or their parents are displayed.

When enabled and no objects are selected, or selection is cleared, the full model is shown in the browser.

The state of the **Show selected** mode is remembered between sessions, and is the same for all opened documents.

18.17.8 Mechanical entities list

Displays a hierarchical tree list of mechanical entities, such as user parameters, constraints, parts, arrays and features. Top-level nodes can have sub-nodes. For example, constraint node have sub-node(s) for its argument(s). An array node has sub-nodes for its arguments, etc.

Double click a node to expand or collapse it, double click (click-pause-click) a node to edit its label if applicable. Not all node names can be edited.

Each node in the list has associated context menu. Right-click a node to invoke the menu. The content of the menu can be different for each kind of node. You can also select a group of nodes and invoke a context menu for them. The menu will contain only those elements which are applicable to each selected node.

Context menu for the root component contains the following items:

Update
Updates the hierarchy of mechanical components for the current drawing in case the 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.

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.

Add new parameter
Creates a new parameter in the assembly.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

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 sequence
Creates a block with a representation of the current assembly.

Renumerate child nodes
Renumerates all child nodes according to their types.

Renumerate child nodes continuously
Renumerates all child nodes with continuous numeration when applicable.

Collapse all
Collapses the main component and all components and subcomponents.

Expand all
Expands the main component and all components and subcomponents.

Context menu for child (non-root) components, internal or external, also offers:

Open
Opens the referenced drawing (see the BMOPEN command).

Open a copy
Opens a copy of a component insert as a new drawing (see the BMOPENCOPY command).

Update
Reloads all referenced components from external files and updates BOM tables (see the BMUPDATE command).

Replace…
Replaces a component insert (see the BMREPLACE command).

Note: Replacing a local insert turns it into an external insert.

Replace all inserts...
Replaces all inserts that refer to the same source (see the BMREPLACE command).

Switch to local
Switches an external component to an internal component (see the BMLOCALIZE command).

Switch to external
Switches an internal component to an external component (see the BMEXTERNALIZE command).

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, the design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

Reset instance custom properties
Resets instance custom properties to defaults inherited from a corresponding block.

Hide / Show
Hides / Shows the component.

Visual Style
Sets a visual style for the component.

Zoom to
Zooms to the selection set.

Select
Selects the component.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same definition and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlights all components with the same definition and the same parameter values.

Delete
Analogue to the SMDELETE command. In this case, the feature is removed from the browser and the geometry is changed depending on the type of the feature.

Set material to component
Opens the **Physical Materials** dialog box, which allows you to assign a physical material to the component.

Remove material from component...
Removes the physical material definition from a local component.

Make component standard / Make component non-standard
Changes the type of the component.

Include all inserts to section
Sets the **Sectionable** property of all similar inserts to **YES**. Defines whether an insert is affected by the VIEWSECTION command.

Exclude all inserts to section
Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the VIEWSECTION command.

Renumerate similar nodes
Renumerates all nodes of the same type and the same level.

Component's BOM status
Controls the appearance of the component in BOM tables.

Context menu for Parameters contains additional tools:

Geometry-driven
If ticked, it makes the parameter geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Context menu for Component parameters contains additional tools:

Link to parameter
Links subcomponent parameter to the main level parameter.

Context menu for Constraints contains additional tools:

Enabled
Toggles the enabling of the constraint.

Flip side
Flips the side of the constraint (not available for fix constraints).

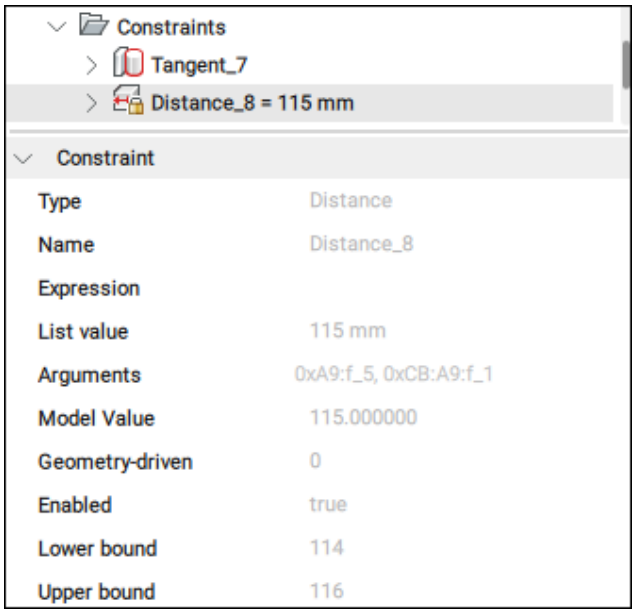
Select geometry
Selects the entities involved in the constraint.

Context menu for dimensional Constraints contains additional tools:

Geometry-driven
If ticked, it makes the constraint geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.



Create design table
Creates a design table to drive parametric block parameters (see the **Parameters Manager** panel).

Context menu for constraint arguments contains additional tools:

Flip
Reverses direction.

Replace...
Replaces the current argument with a another one (see the DMCONSTRAINT3D command).

Exclude
Excludes the current argument.

18.17.9 Properties

Displays the properties of the selected item(s). If several nodes are selected in the tree, the Properties section displays the properties of all the selected nodes. Changing a property modifies all the selected nodes.

Change a property by clicking a numeric field to edit its value or by selecting another option from the drop-down list, which is opened when you press the down arrow from the right side of the field.

The most important properties of a mechanical block are:

- **Extension type**: defines the extension type for the mechanical block.
- **Sectionable**: defines if the mechanical block is sectionable.
- **Standard component**: defines if the mechanical block is a standard component.
- **BOM status**: defines the BOM status.
- **Material**: defines the material.

  Note: Click the browse button (…) to open the **Physical Materials** dialog box.

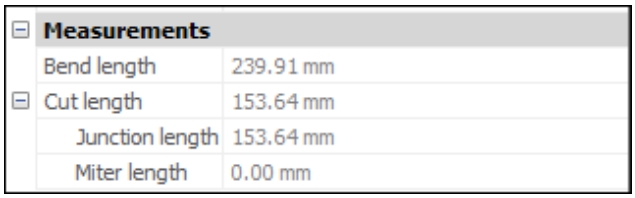
- **Custom properties**: displays the custom properties.

Note:

- Click the browse button (…) to open the **Mechanical Properties** dialog box.
- Click the **Clear material** button to remove the material.

- **Preferable direction**: indicates the direction for exploded views, which is set with the BMEXPLODECONFIG command.

Note: Select a Sheet Metal Body to display the **Measurements** properties in the **Mechanical Browser**.



18.18 MENU command

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

Shape Lite Pro Mechanical BIM

Icon:

18.18.1 Description

Opens the **Choose a Customization File** dialog box to select a CUI, CUIX, MNU, MNS, or ICM file to load. These customization files modify the BricsCAD user interface.

18.19 MENULOAD command

Opens the **Customization Groups** dialog box.

Shape Lite Pro Mechanical BIM

18.19.1 Description

Opens the **Customization Groups** dialog box to load and unload customization groups.

18.20 MENUUNLOAD command

Opens the **Customization Groups** dialog box.

Shape Lite Pro Mechanical BIM

18.20.1 Description

Opens the **Customization Groups** dialog box to load and unload customization groups.

18.21 MIDPOINT command

Toggles the **Midpoint** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



18.21.1 Description

Toggles the **Midpoint** entity snap to enable or disable snapping to midpoint. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

18.22 MINSERT command

Inserts a block as a rectangular array.

Shape Lite Pro Mechanical BIM

18.22.1 Description

Inserts a block as a rectangular array. It is a combination of the –INSERT and -ARRAY commands and creates a MInsert Block entity.

18.22.2 Options within the command

Block to insert
Specify the name of the block to be inserted.

~
Displays the Insert Block dialog box.

Insertion point for block
Specify the insertion point of the selected block.

Scale
Allows to set the X and Y scale factor equally.

Corner
Specifies the size of the block by picking a second point.

X scale
Specifies the scale factor of the block in the x direction.

Y scale
Specifies the scale factor of the block in the y direction.

Z scale
Specifies the scale factor of the block in the z direction.

Rotate
Specifies the rotation angle of the block around its insertion point.

Multiple blocks
Allows to insert a rectangular array of multiple blocks.

Number of rows in the array
Specifies the number of rows.

Note: To insert a linear array, enter 1. To draw the array downwards, enter a negative number.

Number of columns in the array
Specifies the number of columns.

18.23 MIRROR command

Mirrors entities about a mirror line in a 2D plane.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: MI

Note: MIRRTEXT system variable determines whether text is mirrored by the MIRROR command or not.

18.23.1 Description

The entities are mirrored about a mirror line, which you define by specifying two points.

Note: To mirror vertically or horizontally, press the Shift-key when specifying the second point or use Polar Tracking.

18.23.2 Options within the command

Delete the original entities?
Determines whether the original entities are erased or not.

18.24 MIRROR3D command

Mirrors entities about a mirror plane in 3D space.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: 3DMIRROR, 3DM

18.24.1 Description

Creates a mirror copy of selected entities in three-dimensional space.

18.24.2 Options within the command

Entity
Defines the mirror plane by extrusion direction.

Note: This option does not work with flat entities or 3D solids. To give an entity an extrusion, use the Properties panel's Thickness option.

Last
Reuses the last mirror plane.

View
Defines the mirror plane as the current view plane.

Z axis
Defines the mirror plane by the z axis and a point on the view plane.

X-Y plane
Specifies the mirror plane as the x,y plane.

Y-Z plane
Specifies the mirror plane as the y,z plane.

Z-X plane
Specifies the mirror plane as the z,x plane.

3 points
Defines the mirror plane by three points.

Delete the original entities?
Determines whether the original entities are erased or not.

18.25 MKLTYPE command (Express Tools)

Creates a linetype based on selected entities.

Shape Lite Pro Mechanical BIM

Icon:

18.25.1 Method

Opens the **MKLTYPE - Select Linetype file to create** dialog box that allows you to save the linetype definition.

To create the linetype definition specify the linetype name, a description (optional), and the start point and end point.

18.25.2 Options within the command

Specify linetype name
Specify a name for the linetype.

Enter (optional) description for linetype
Type a description for the linetype.

Note: Press Enter to leave this field blank.

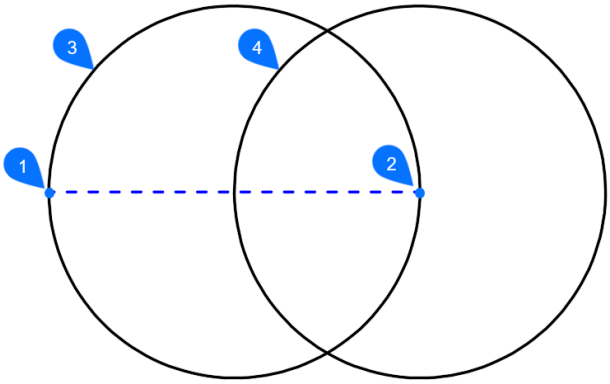
Specify start point for linetype definition
Allows you to define the starting point for the linetype definition (1).

Specify end point for linetype definition
Allows you to define the ending point for the linetype definition (2).

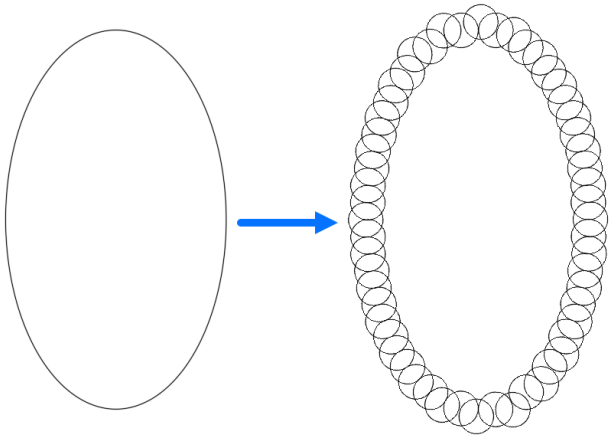
Select entities
Allows you to select the entities you want to use (3 and 4).

Note: Supported entities: point, line, polyline, text, shape entities (created with MKSHAPE command).

In the following image there are illustrated two shape entities.



The linetype is automatically loaded in the current drawing.



18.26 MKSHAPE command (Express Tools)

Creates a shape definition based on selecting entities.

Shape | Lite | Pro | Mechanical | BIM

Icon:

18.26.1 Method

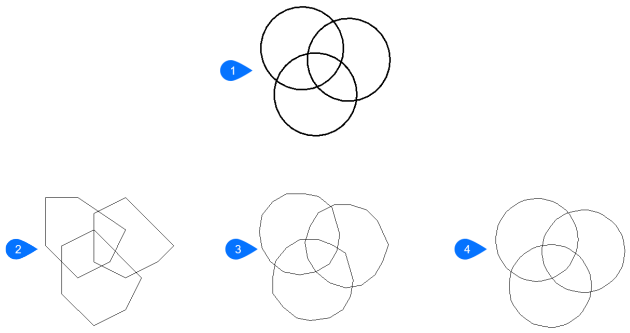
Opens the **MKSHAPE - Select the Shape File to create** dialog box that allows you save the shape definition.

To create the shape definition:

1 Specify the shape name.

2 Enter the resolution. The larger this value, the more accurate the representation of the selected geometry will be, but the more processing time will be needed.

3 ***Note:*** The resolution value will be round off to a multiple of 8 and the maximum value equals 32,767.



- 1. Original objects
- 2. Shape resolution = 8
- 3. Shape resolution = 48
- 4. Shape resolution = 160

4 Specify a base point for the shape.

5 Select the entities you want to use and press enter. A shape will be created.

Note: The created shape can be inserted in your drawing using the SHAPE command.

Note: The created shape can be loaded in an other drawing using the LOAD command.

18.27 MLEADER command

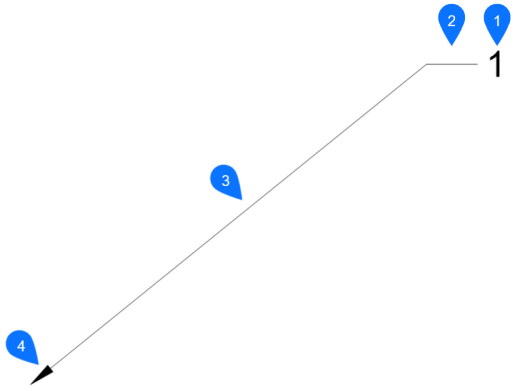
Creates multileader entities using the current multileader style.

Shape Lite Pro Mechanical BIM

Icon:

18.27.1 Description

Creates a multileader with three components: a leader line with an optional arrowhead, a landing line and the content, such as text or a block. Multileaders support more than one leader line per content.



1 Mtext/block

2 Landing

3 Segment

4 Arrowhead

18.27.2 Method

There are three methods to begin creating a multileader:

- leader arrowHead first
- leader Landing first
- Content first

18.27.3 Options with the command

leader arrowHead first
Create a multileader by first specifying the location of the arrowhead.

leader Landing first
Create a multileader by first specifying the location of the leader landing.

Content first
Create a multileader by first specifying the location of the content.

Leader type
Sets the leader type: straight, spline or none.

Straight
The leader is drawn from straight segments.

SPline
A spline is drawn using the pick points as the control points.

None
No leader is drawn.

leader lAnding
Specifies whether to draw a landing or not.

Content type
Specifies the content type of the multileader.

Block
Uses the block name you insert.

Mtext
Uses mtext; displays the MText editor.

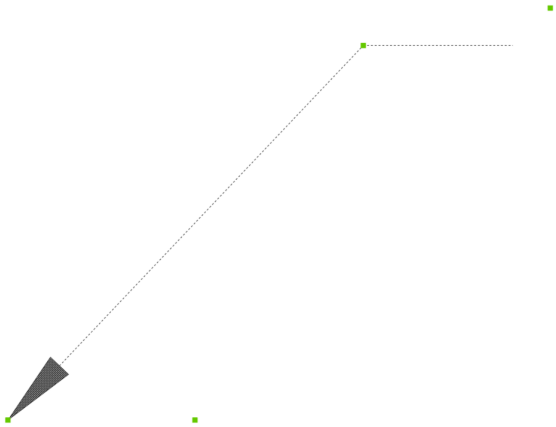
None
Draws no content.

Maxpoints
Enter the maximum points.

First angle
Enter the first angle constraint.

Second angle
Enter the second angle constraint.

Grip editing
By clicking the green squares, you can edit the grip points of the multileader.



18.28 MLEADERALIGN command

Aligns the balloons of two or more multileaders.

Shape Lite Pro Mechanical BIM

Icon:

18.28.1 Description

Aligns the balloons of two or more multileaders to each other, to a polyline or array around a circle. You can also specify the spacing between multileaders.

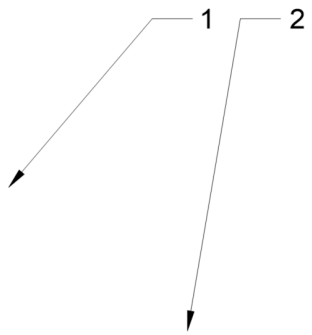
18.28.2 Method

This command offers five methods to align multileaders:

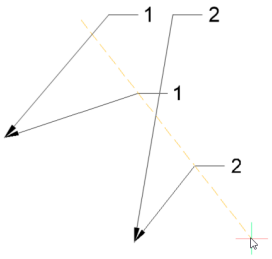
- byMleader
- onpolyLine
- Parallel
- Spacing
- Circle

18.28.3 Options within the command

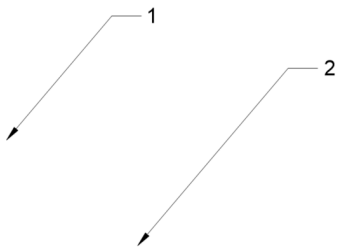
byMleader
Align the landings of the multileaders to the landing of a selected multileader.



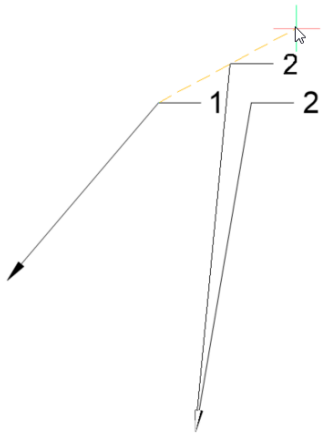
onpolyLine
Align multileaders along an imaginary polyline.



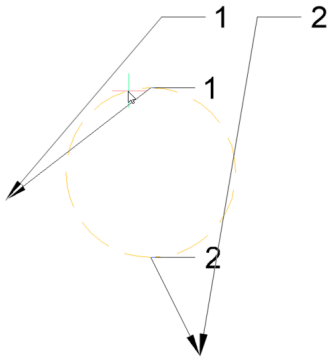
Parallel
Align the segment of the multileaders parallel to the segment of a selected multileader.



Spacing
Specify the spacing between multileaders.



Circle
Arrays the multileaders around a circle.



18.29 MLEADERCOLLECT command

Collects two or more multiline leaders into a single leader.

Shape Lite Pro Mechanical BIM

Icon:

18.29.1 Description

Collects two or more multiline leaders into a single leader in a horizontal or vertical arrangement. The leaders can be sorted alphabetically.

Note: The command only works with mleaders that have blocks as content.

18.29.2 Method

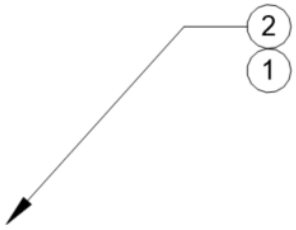
There are five methods to collect multileaders:

- Vertical
- Horizontal

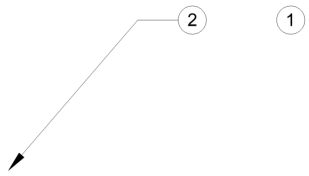
- Wrap
- Sorting
- Collapsing

18.29.3 Options within the command

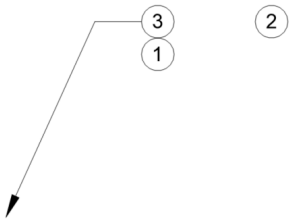
Vertical
Arranges the content blocks vertically to a single leader.



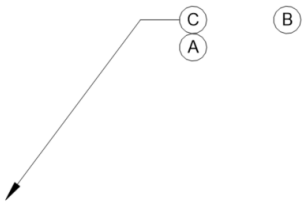
Horizontal
Arranges the content blocks horizontally to a single leader.



Wrap
Arranges the content blocks horizontally to a maximum width, and then places the remaining blocks in the row below.



Sorting
Sorts the content by name.



None
Does not sort the blocks.

Ascending
Sorts from A to Z.

Descending
Sorts from Z to A.

Collapsing
Arranges the content blocks horizontally to a single leader.



18.30 MLEADEREDIT command

Edit leaders from multileader entities.

Shape Lite Pro Mechanical BIM

Icon:

18.30.1 Description

Adds and removes leaders to and from multileader entities.

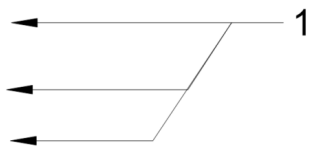
18.30.2 Method

There are two methods to edit multileaders:

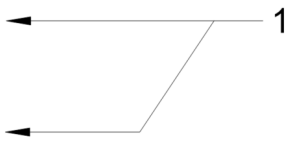
- Add leaders
- Remove leaders

18.30.3 Options within the command

Add leaders
Adds leaders to the selected mleader entity.



Remove leaders
Removes leaders from the selected mleader entity.



18.31 MLEADEREDITEXT command

Edit leaders from multileader entities.

Shape Lite Pro Mechanical BIM

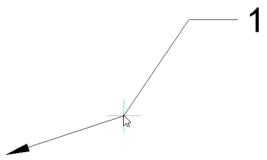
18.31.1 Description

Adds and removes leader lines, adds and removes vertices, and changes the landing length of a multileader entity (short for "multiline leader edit extended").

18.31.2 Options within the command

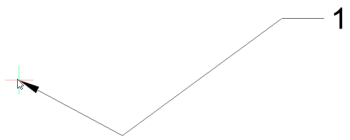
Add vertex

Adds a vertex to the leader line.



add vertex at ENd

Adds a vertex and a new leader line at the arrowhead.

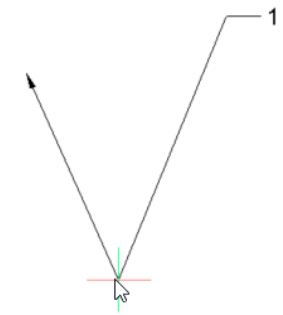


Remove vertex

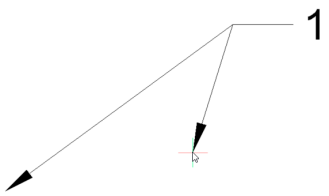
Removes a vertex from the leader line.

Stretch vertex

Stretch a leader line by moving a vertex.



add Leader
Adds one or more leader lines to an existing mleader.



rEmove Leader
Removes a leader and associated vertex from the leader line.

lengthen LAnding
Changes the length of the landing.



18.32 MLEADERSTYLE command

Opens the **Drawing Explorer** dialog box with **Multileader Styles** selected.

Shape Lite Pro Mechanical BIM

18.32.1 Description

Opens the **Drawing Explorer** dialog box with **Multileader Styles** selected to view and modify multileader styles in the current drawing.

18.33 MLINE command

Creates an MLine.

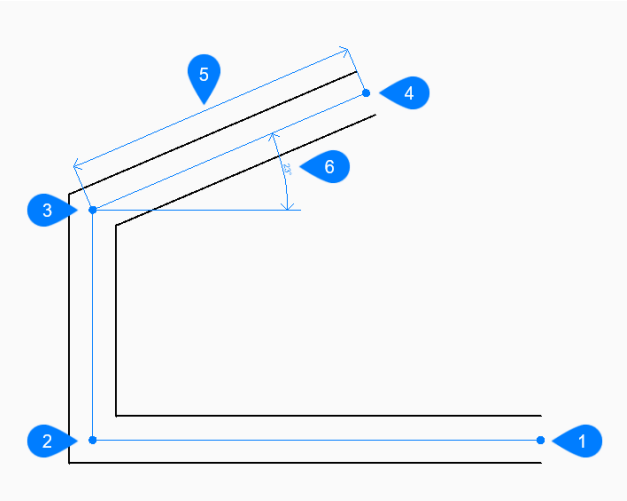
Shape Lite Pro Mechanical BIM

Icon:

Alias: ML

18.33.1 Description

Creates a single MLine entity with multiple parallel lines and segments by specifying the start and endpoint of each segment. The MLine is based on the current multiline style. Options allow you to specify the justification and scale.



1 Start

2 Next

3 Next

4 End

5 Length

6 Angle

18.33.2 Method

This command has 3 methods to begin creating an MLine:

- Start of line
- Last point
- Follow

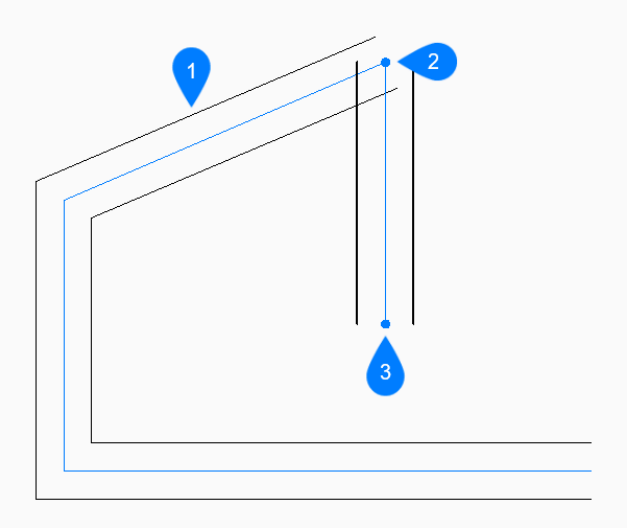
18.33.3 Options within the command

Start of line
Allows you to begin creating an MLine by specifying the start point.

Set end point
Specifies the endpoint of the MLine segment.

Note: You can continue adding unlimited segments until you press **Enter** key to end the command.

Last point
Begins creating an MLine from the last point picked.

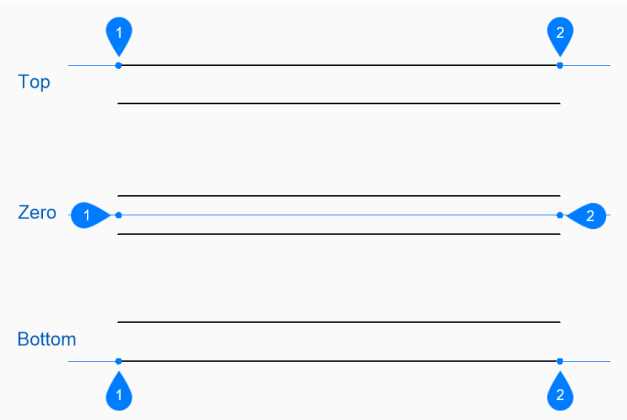


1 Last drawn segment

2 Last point

3 End point

Justification
Specifies the justification of the MLine relative to the start point.



1 Start

2 End

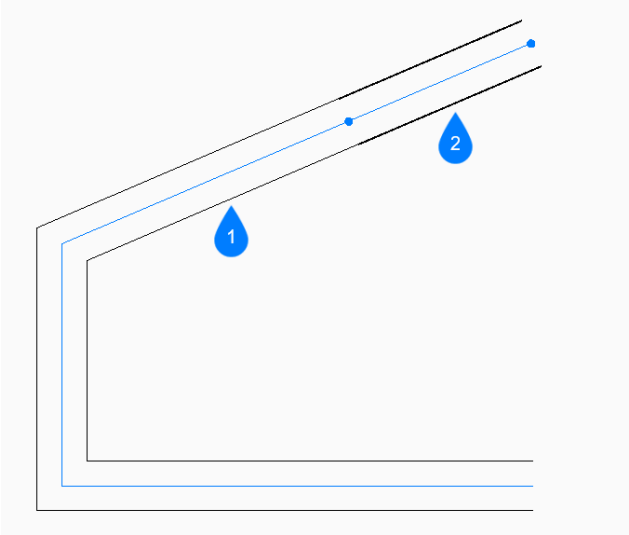
The justification is saved in the CMLJUST variable.

Scale
Specifies the overall scale of the MLine.
The scale is saved in the CMLSCALE variable.

Style
Specifies the style to use for the MLine as defined by the MLSTYLE command. The style is saved in the CMLSTYLE variable.

Follow
Begins creating an MLine from the last drawn line segment, following its angle.

Length of line
Specifies the length of the MLine segment. Picking a point only determines the length since the angle follows the previous segment.



1 Last drawn segment

2 Follow

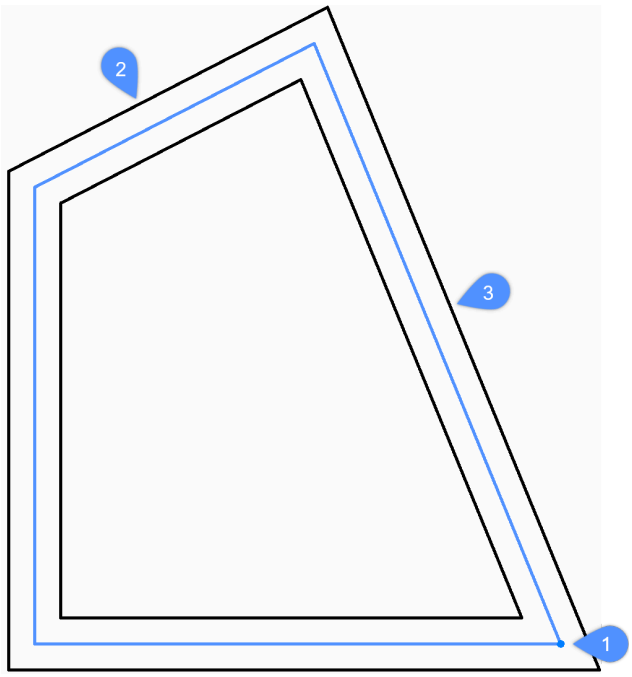
Angle
Specifies the angle of the MLine segment measured counterclockwise from the positive x-axis.

Length
Specifies the length of the MLine segment.

Angle of line
Specifies the angle of the MLine segment measured counterclockwise from the positive x-axis.

Undo
Undoes the last MLine segment and continue drawing from its previous start point.

Close
Automatically draw an MLine segment from the end point of the last segment to the start point of the first segment.



1 Start/End

2 Last drawn segment

3 Close

18.34 MLSTYLE command

Opens the **Drawing explorer** dialog box with **Multiline Styles** selected.

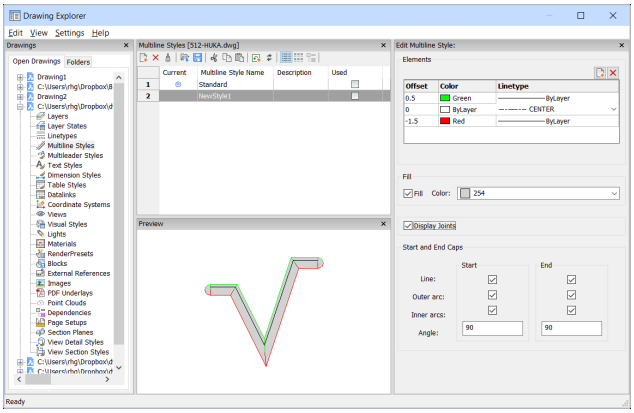
Shape Lite Pro Mechanical BIM

Icon:

18.34.1 Description

Opens the **Drawing explorer** dialog box with Multiline Styles category selected to view and modify multiline styles in the current drawing.

Note: Some properties defined by this command can be overridden by options in the MLINE command.



18.34.2 Options within the Edit Multiline Style panel

Elements
Edits the elements of the multiline.

Add
Adds a line to the multiline definition using the default properties.

- Offset = 0 drawing units
- Color = Bylayer
- Linetype = Bylayer

Delete
Erases a line from the multiline definition without warning.

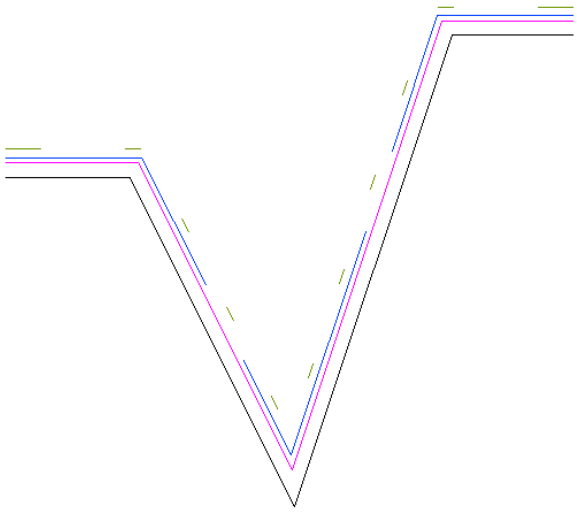
Note: Following multiline style cannot be deleted: The Standard Style.

Offset
Specifies how far each line is from the center line of the mline.

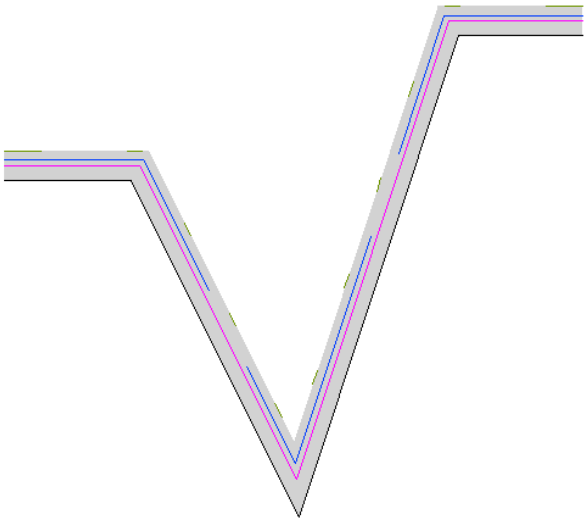
Color
Specifies the color of each line.

Linetype
Specifies the linetype for each line. You can choose a linetype from the list or choose load to access additional linetypes.

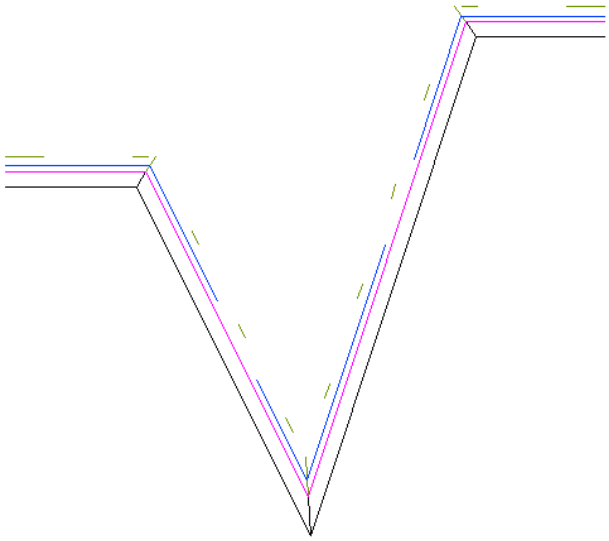
Note: As you add elements and change properties, the preview pane shows what the mline now looks like. If the changes do not load, you should click regen or close and reopen the dialog box.



Fill
When turned on, a solid fill floods the mline between the outer lines.

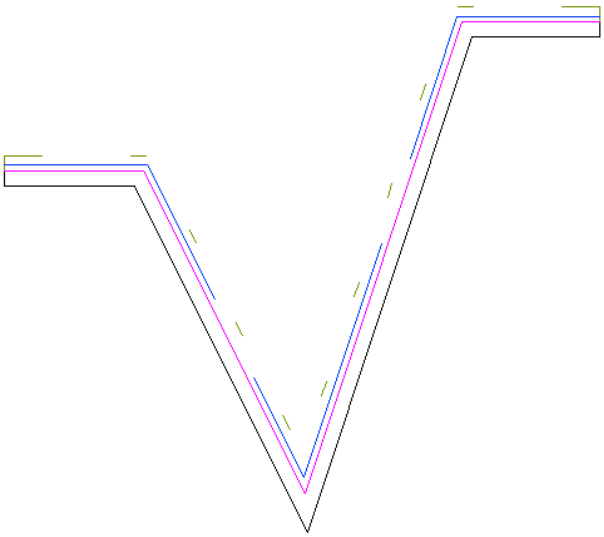


Display Joints
When selected, the mline draws joints (diagonal lines) between segments.

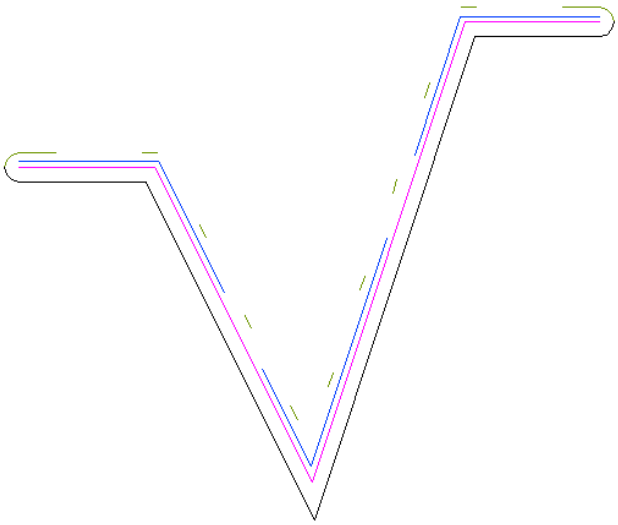


Start and End Caps
Closes the end segments of open multilines by a variety of methods.

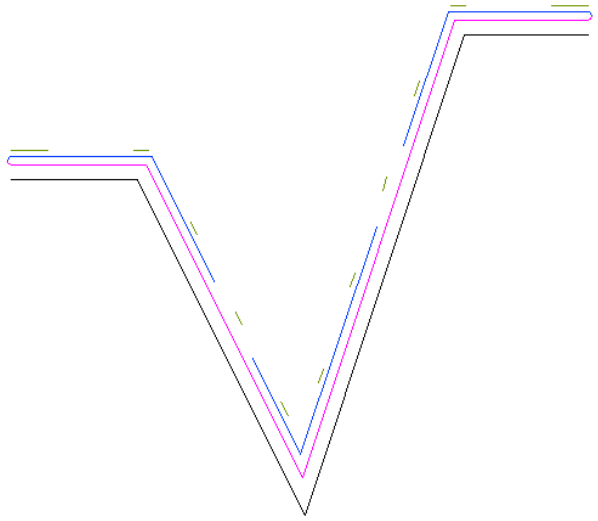
Line
Draws a straight line to close the end segments.



Outer arc
Draws an arc which connect the outer lines of the multiline to close the end segments.



Inner arcs
Draws arcs which connect the inner lines of the multiline to close the end segments.



Angle
Sets the angle of the outer and inner arcs.

18.34.3 Context Menu Options

New
Creates a new multiline style definition. Displays the **New Multiline Style** dialog box (see **New Multiline Style dialog box** related article).

Delete
Deletes multiline style definitions from the drawing. The following multiline style definitions cannot be deleted:

- Standard style
- The styles in use

Load from mln file
Loads multiline styles from a multiline (*.mln) file. Displays the **Load Multiline Styles** dialog box (see **Load Multiline Style dialog box** related article).

Save to mln file
Saves all multiline styles in the drawing to a multiline (*.mln) file for use by other CAD systems. Displays the **Save Multiline Styles** file dialog box.

Rename
Renames the multiline style.

Note: Following multiline style cannot be renamed: Standard style.

Select All
Selects all multiline style definitions.

Invert selection
Deselects the current selection and vice versa.

18.35 MOCORO command (Express Tools)

Moves, copies, rotates or scales entities.

Shape Lite Pro Mechanical BIM

Icon: 

18.35.1 Method

Select the entities and specify a base point.

18.35.2 Options within the command

Move
Allows you to move the selected entities.

Copy
Allows you to make a copy of the selected entities.

Rotate
Allows you to rotate the selected entities around the specified base point.

Scale
Allows you to scale the entities.

Base
Allows you to select a new base point.

Undo
Undoes the last change.

18.36 MODELERPROPERTIES command

Opens the **Settings** dialog box with the **Modeler properties for ACIS** category expanded.

Shape Lite Pro Mechanical BIM

18.36.1 Description

Opens the **Settings** dialog box with the **Modeler properties for ACIS** category expanded to view and modify relevant system variables.

18.37 -MODELERPROPERTIES command

ACIS modeler settings.

Shape Lite Pro Mechanical BIM

18.37.1 Description

Specifies settings for the ACIS modeler.

18.37.2 Method

There are 14 methods to set ACIS modeling properties:

- view Properties

- USe facetres
- Surface tolerance
- Normal tolerance
- Max grid lines
- Edge length
- grid aspect Ratio
- Adjust mode
- Grid mode
- Triangulation mode
- min U grid lines
- min V grid lines
- mass properties aCcuracy
- check Level

Note: Turning on FACETRES turns off all other settings.

To adjust the Hidden Line Accuracy setting, use the MODELPROPERTIES command.

18.37.3 Options within the command

view Properties
Reports the value of each setting.

USe facetres
Reads the value of the FACETRES variable to set the smoothness of shaded and rendered scenes.

Surface tolerance
Sets the value for the SPANORMALTOL variable.

Normal tolerance
Sets the value for the SPASURFACETOL variable.

Max grid lines
Sets the value for the SPAMAXNUMGRIDLINES variable.

Edge length
Sets the value for the SPAMAXFACETEDGELENGTH variable.

grid aspect Ratio
Sets the value for the SPAGRIDASPECTRATIO variable.

Adjust mode
Sets the value for the SPAADJUSTMODE variable.

Grid mode
Sets the value for the SPAGRIDMODE variable.

Triangulation mode
Sets the value for the SPATRIANGMODE variable.

min U grid lines
Sets the value for the SPAMINUGRIDLINES variable.

min V grid lines
Sets the value for the SPAMINVGRIDLINES variable.

mass properties aCcuracy
Sets the value for the MASSPROPACCURACY variable.

check Level
Sets the value for the SPACHECKLEVEL variable.

18.38 MOVE command

Moves entities.

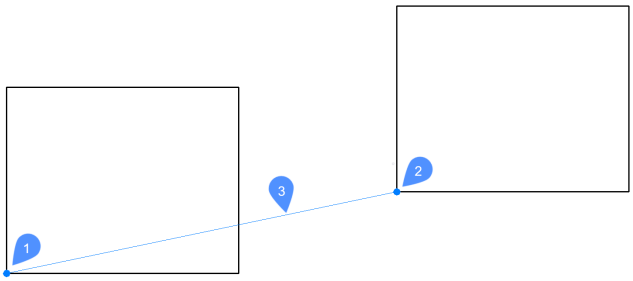
Shape Lite Pro Mechanical BIM

Icon:

Alias: M

18.38.1 Description

Moves entities by specifying the starting point and the destination.



1 Base point

2 Second point

3 Displacement distance

18.38.2 Options within the command

Displacement
Specify the displacement vector (the distance at which to position the copy). 'Vector' means that you specify the distance and angle at the same time.

Note: When Dynamic Input Mode is active, you can type a distance and angle in the dynamic input fields.

18.39 MOVEBAK command (Express Tools)

Sets the destination folder for backup files.

Shape Lite Pro Mechanical BIM

Icon:

18.39.1 Method

Specify a new folder name for all BAK files.

Note:

- Enter a . (dot) to clear the MOVEBAK folder name. New BAK files will be created in the original folder.
- Type ~ to open the **Select Folder** dialog box, which allows you to select the MOVEBAK directory.
- The creation of backup files is controlled by ISAVEBAK system variable.

18.40 MOVEEDATA command

Moves extended entity data from one entity to one or more others.

Shape Lite Pro Mechanical BIM

Icon:

18.40.1 Method

Enter the name of the application to which the entity data belongs, select the entity from which to move Entity Data and one or more entities to which Entity Data will be moved (that will receive the edata). Extended Entity Data can be created with EDITEDATA command.

18.40.2 Options within the command

? to list application names
Lists names of applications loaded in the current drawing.

18.41 MOVEGUIDED command

Moves entities using guide curves.

Shape Lite Pro Mechanical BIM

Icon:

18.41.1 Description

Automatically aligns moved entities with relevant geometry, using temporary guide curves. Polyline segments are also accepted as guide lines.

18.41.2 Method

The command can be executed in two ways:

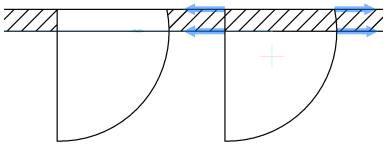
Pre-pick mode
First select entities, then launch the command.

Note: The entities to be copied are displayed in green.

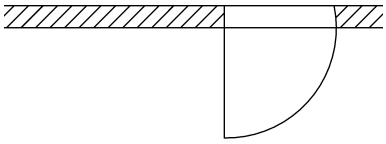
Post-pick mode
Launch the command, then select the entities using a selection window.

Note: All entities that fall completely within the selection window, including polyline segments, are included in the copy selection set and are displayed in yellow. Entities that cross the selection window are used as guide curves and are displayed in blue.

Blue arrows indicate the anchor points and direction of the guide curves. The moved entities will only align with geometry that matches the number of guide curves and distances between them.



Click to place the moved entity or type a distance in the dynamic entry fields.



18.41.3 Options within the command

Polygonal
Creates a polygonal selection window.

Rectangular
Creates a rectangular selection window.

Region
Pastes the clipped area inside the selection window.

Entities
Pastes the entities inside the selection window.

Save detail
Allows to save a detail as a block/Parametric block.

Note: Displays the **Write block to file** dialog box. See the related article **Write block to file dialog box**.

Flip entities
Flips the entities along a line perpendicular to the reference curves.

Note: You can also press the Shift key to flip the entities.

choose base Point
Defines a base point for insertion.

use Center point
Uses the center point for insertion.

18.42 MPEDIT command (Express Tools)

Edits multiple polylines and converts lines and arcs into polylines.

Shape Lite Pro Mechanical BIM

18.42.1 Method

The MPEDIT command is similar to the PEDIT command, but it has the ability to operate on multiple polylines at once.

18.42.2 Options within the command

Convert Lines and Arcs to polylines?
Allows you to decide if the selected lines and arcs are converted to polylines.

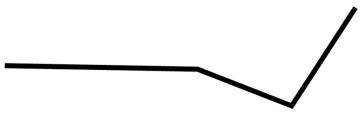
Open
Opens closed polylines by removing the last segment drawn to close the polyline.

Close
Closes the polyline by adding a segment between the start and endpoints.

Join
Adds open entities to the selected polyline. A fuzzy distance has to be specified.

> ***Note:*** This option works only with open and with cojoined entities. Cojoined means that the endpoints of entities meet, such as when drawn with ENDpoint object snap or with the Last point option. The joined entities take on the source polyline's properties, such as color, width, and layer.

Width
Changes the width of all segments.



> ***Note:*** This option overrides tapered widths.

Fit
Fits a curve to the polyline.

> ***Note:*** To unfit the polyline, use the **Decurve** option.

Spline
Converts the polyline into a spline.

> ***Note:*** All width information is lost. Use the **Width** option to reapply width. The spline is a Bezier-spline whose smoothing is defined by the SPLINETYPE system variable.

Decurve
Reverses the effects of the **Fit** and **Spline** options.

Ltype gen
Determines how linetypes appear on polylines.

ON
Linetypes start and stop at the polyline's start and endpoints.

OFF
Linetypes start and stop at each vertex.

Note: This option is stored in the PLINEGEN system variable.

Undo
Undoes the last action.

18.43 MSLIDE command

Opens the **Create Snapshot** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: MSNAPSHOT

18.43.1 Description

Opens the **Create Snapshot** dialog box to save data from the current drawing to a SLD file.

18.44 MSPACE command

Switches from paper space to model space viewports.

Shape Lite Pro Mechanical BIM

Alias: MS

18.44.1 Description

Switches from paper space to model space viewport in a layout tab. This command has a shortcut menu that opens by right-clicking on **M:Layout** on the Status Bar.

Note: This command operates only when a layout tab is in paper space mode. To reverse the action of this command double-click outside the viewport border, or use PSPACE command.

18.45 MSTRETCH command (Express Tools)

Stretches entities by defining multiple crossing windows or crossing polygons.

Shape Lite Pro Mechanical BIM

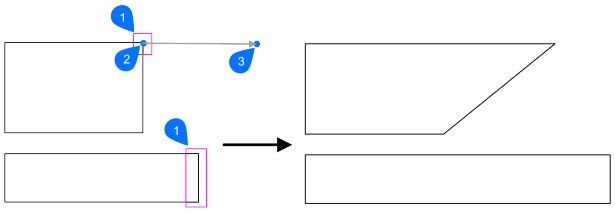
Icon: 

18.45.1 Method

1 Define crossing windows or polygons (1).

2 Specify a base point (2).

3 Specify a second base point (3).



18.45.2 Options within the command

CP
Creates a crossing polygon to stretch all entities touched by it.

Note: Right-click to close the polygon defining.

C
Creates a crossing window to stretch all entities inside it.

Done
Exits the command if no selections have been made.
If selections have been made, it continues the command.

Undo
Removes the last defined window for stretch.

Specify base point
Allows you to pick a point or enter the coordinates to specify the base point.

display
The command uses the lower-left extent of the current view as the base point.

Remove objects
Removes entities from the selection set.

Specify second point
Allows you to pick a point or enter the coordinates to specify the second point.

18.46 MTEXT command

Places text in a bounding box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: MT, T

18.46.1 Description

Places formatted paragraph text in a bounding box that limits the extent of the text.

After specifying the limits of the text bounding box, the **Text Formatting** toolbar opens.

You can edit Mtext bounding boxes directly using grips:



1 Drag to change the width of the bounding box.

2 Drag to change the height of the bounding box.

18.46.2 Options within the command

Justification
Specifies the justification (horizontal alignment) of the text inside the bounding box.

Rotation angle
Specifies the rotation angle of the text block. Positive angles rotate the text block counter-clockwise.

text Style
Specifies the text style to use. See the Style command.

text Height
Specifies the height of the text.

Note: This option is unavailable when the style specifies the text height.

Direction
Specifies the direction in which the bounding box is expanded.

Left-to-right
The text is placed on the left and expands the bounding box to the right.

Top-to-bottom
The text is placed at the top and expands the bottom of the bounding box.

By-Style
Uses the direction defined by the text style.

Width
Specifies the width of the bounding box.

Note: When set to 0, the box has an *infinite* width.

Line spacing
Defines the text's line spacing style and line spacing factor.

Note: These settings default value is specified by the TSPACETYPE and TSPACEFAC system variables.

At least
Specifies the minimum line spacing factor.

Exactly
Specifies the exact line spacing factor.

Columns
Defines the amount of columns.

No columns
No columns are created.

Static
A fixed amount of columns with fixed dimensions is created.

Dynamic
A dynamic amount of columns is created. The number of columns depends on the length of the text.

18.47 -MTEXT command

Creates multiline text with the Command line.

Shape Lite Pro Mechanical BIM

Places formatted, paragraph text in a bounding box that limits the extent of the text.

Note: The text does not appear in the drawing until you end the command.

18.47.1 Description

Creates a multiline text by specifying the first corner and the opposite corner for block of text.

18.47.2 Options within the command

Justification
Specifies the horizontal alignment of the text inside the bounding box. Choose between: Top-Left, Top-Center, Top-Right, Middle-Left, Middle-Center, Middle-Right, Bottom-Left, Bottom-Center, Bottom-Right.

Rotation angle
Specifies the rotation angle of the text block.

Note: Positive angles rotate the text block counter-clockwise.

text Style
Specifies the text style to use for the mtext.

?
Lists all styles defined in the current drawing.

text Height
Specifies the height of the text.

Note: This option is unavailable when the style specifies the text height.

Direction
Specifies the direction in which the bounding box is expanded.

Left-to-right
The text is placed at the left and expands the bounding box to right.

Top-to-bottom
The text is placed at the top and expands the bottom of the bounding box.

by-Style
Uses the direction defined by the text style.

Width
Specifies the width of the bounding box.

Note: When set to 0, the box has "infinite" width.

Line spacing
Specifies the line spacing of the mtext.

At least
The lines of text will be automatically adjusted, based on the highest character in the line.

Exactly
The line spacing will be the same for all lines of the mtext.

Columns
Specify the properties of the columns.

No columns
Sets no column to the mtext.

Static
Specify total width, number of columns, gutter (space between columns) and height of the columns.

Dynamic
Specify width, gutter (space between columns) and height of the columns.

18.48 MTP command

Allows to specify a point as the midpoint.

Shape Lite Pro Mechanical BIM

18.48.1 Description

Allows to specify a point as the midpoint between two points whenever a command prompts to enter a point.

Note: This is a transparent command.

18.48.2 Method

This command works only with commands that prompts to enter a point.

Note: This command sets entity snaps, similar to OSNAP command.

18.48.3 Options within the command

First point of mid
Identify the first reference point.

Second point of mid
Identify the second reference point.

18.49 MULTIPLE command

Repeats commands.

Shape Lite Pro Mechanical BIM

18.49.1 Description

Repeats commands until you press the **Esc** key.

Note: This command is useful for commands that do not repeat automatically.

18.50 MVIEW command

Creates one or more viewports in Paper space (short for "make viewports") to display entities drawn in Model space. Each viewport can have its own settings, shown below.

Shape Lite Pro Mechanical BIM

Icon:

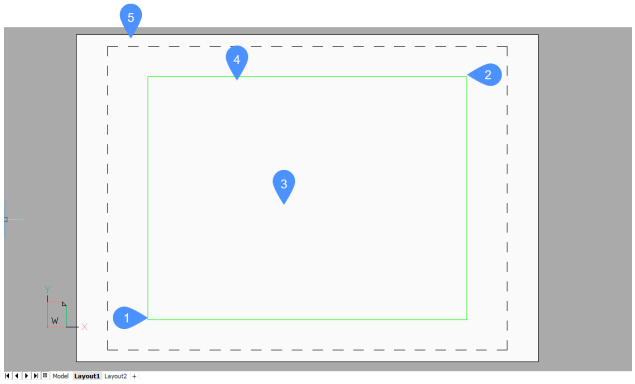
Alias: MV

Note: This command operates only in Paper Space.

Note: To create viewports in Model space, use the VPORTS command.

18.50.1 Description

Specifies the first and the opposite corners to create a rectangular viewport.



1 First corner

2 Opposite corner

3 Viewport

4 Viewport border

5 Paper space (Layout)

18.50.2 Options within the command

ON

Turns on viewports that are turned off.

OFf

Turns off viewports.

This option hides the content of the viewport; the viewport border remains visible. To hide the viewport border, place it on a separate layer, and then freeze the layer.

Lock
Locks the viewport's scale factor.
Set the scale factor with the ZOOM command's nXP option.

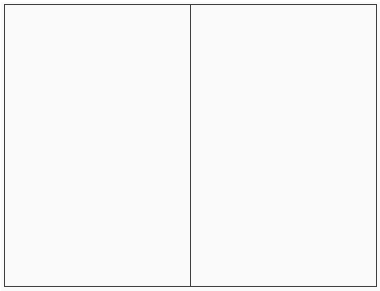
Fit
Draws a rectangular viewport that fits the current layout.

create 2 viewports
Draws two rectangular viewports.

Horizontal
Draws two horizontal viewports of equal size.



Vertical
Draws two vertical viewports of equal size.



create 3 viewports
Draws three rectangular viewports.

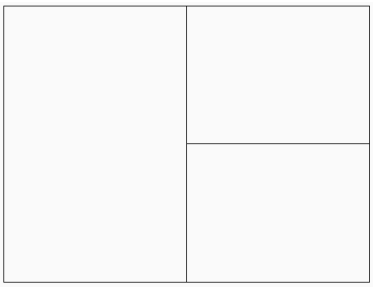
Above
Draws one viewport above two side-by-side viewports.



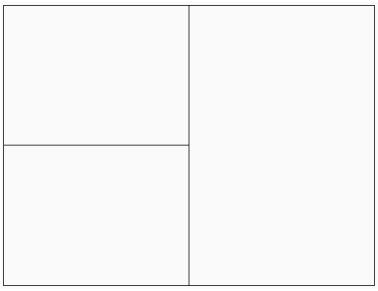
Below
Draws one viewport below two side-by-side viewports.



Left
Draws one viewport to the left of two stacked viewports.



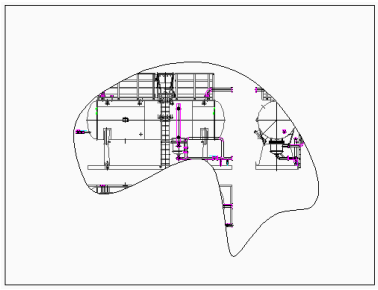
Right
Draws one viewport to the right of two stacked viewports.



create 4 viewports
Draws four rectangular viewports of equal size.



Object
Converts an object into a viewport border.
The viewport is traced over the object; the original object remains in the drawing. This option is useful for ensuring a viewport border precisely matches an outline.



Polygonal
Draws non-rectangular viewports made of lines and arcs.

draw Arcs
Enters arc drawing mode; options from ARC command are activated.

draw Lines
Enter line drawing mode; options from LINE command are activated.

Distance
Draws the next line segment at a specified distance and angle.

Follow
Draws the next line segment at the same angle.

18.51 MVSETUP command

Creates multiple paper space viewports and edits them.

Shape Lite Pro Mechanical BIM

18.51.1 Description

Creates multiple paper space viewports and then aligns, rotates and scales them.

Note: This command operates differently, depending on whether it starts in Model tab or Layout tab.

Note: This command can be entered transparently.

18.51.2 Method

There are two methods to use MVSETUP command:

- In Model tab.
- In Layout tab.

18.51.3 Options within the command

Yes
Switches to Layout tab.

No
Stay in Model tab.

Align
Pans the display in a viewport to align it with a base point in a reference viewport.

Note: Depending on the relative position of the two viewports, the display of the panned viewport might be shifted outside of the viewport boundary.

Angled
Pans the view in the second viewport is panned over the specified distance and angle.

Horizontal alignment
Pans the view in the second viewport vertically to align the base point and the anchor point horizontally.

Vertical alignment
Pans the view in the second viewport horizontally to align the base point and the anchor point vertically.

Rotate view
Rotates the view in the viewport by the specified angle.

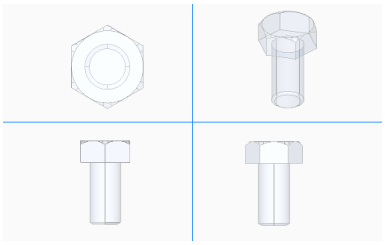
Note: When the VPROTATEASSOC variable is ON, the display in a viewport is rotated if the viewport is rotated.

Undo
Undoes the previous action and returns to the options prompt.

Create
Creates viewports in the current layout.

Delete objects
Deletes viewport entities.

2 - std. engineering
Creates 4 viewports preset to top, front, right and south-east isometric viewpoints.



3 - array of viewports
Creates a desired number of viewports.

Scale viewports
Specifies the scaling inside viewports.

Interactively
Scale each viewport separately.

Uniform
Scale all viewports equally.

Undo
Undoes all operations executed by the current MVSETUP command session.

Scientific
Scales the viewport on a scientific scale.

Decimal
Scales the viewport on a decimal scale.

Engineering
Scales the viewport on an engineering scale.

Architectural
Scales the viewport on an architectural scale.

Metric
Scales the viewport on a metric scale.

19. N

19.1 NAVIGATE command

Changes the way you navigate in BricsCAD.

Shape Lite Pro Mechanical BIM

Icon:

19.1.1 Description

Walks and flies through 3D models with the keyboard, as used in computer games.

19.1.2 Method

There are two methods to navigate through the model:

- Keyboard option: Hover the mouse cursor over the target location and then hold down a key to navigate.
- Mouse option: Hold down left button and drag the mouse.

Right-click to open the **Navigate Settings** dialog box.

19.1.3 Keyboard options

A/Left arrow
Moves to the left.

W/Up arrow
Moves forward.

S/Down arrow
Moves backwards.

D/Right arrow
Moves to the right.

F
Toggles flight mode on and off.

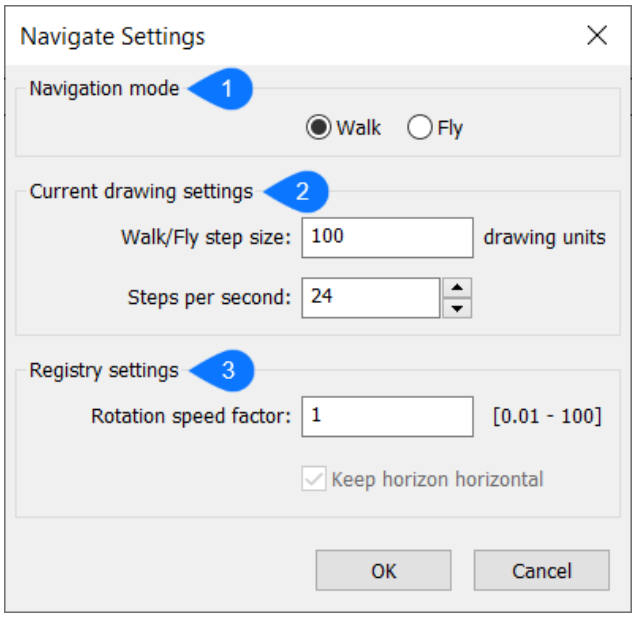
ESC
Exits the command.

19.1.4 Mouse options

Hold down left button and drag the mouse
Look around mode.

Right click
Brings up the **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

19.1.5 Navigation mode

Toggles between Walk and Fly modes:

- Flight mode: permits movements in all three dimensions.
- Walk mode: restricts movement to the xy-plane.

19.1.6 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.

19.1.7 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.

19.2 NCOPY command (Express Tools)

Copies entities nested inside external references and blocks without having to explode or bind it.

Shape Lite Pro Mechanical BIM

Icon: 

19.2.1 Method

1 Select the nested entities you want to copy.

2 Enter a base point.

3 Enter a second point.

19.2.2 Options within the command

Displacement
Allows you to define a displacement vector instead of a second point.

Mode
Toggles between **Single** and **Multiple**.

Multiple
Allows you to make multiple copies.

Array
Asks for a number of copies and a distance.

19.3 NEAREST command

Toggles the **Nearest** entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

19.3.1 Description

Toggles the **Nearest** entity snap to enable or disable snapping to nearest. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

19.4 NETLOAD command

Opens the **Netload** dialog box.

Shape Lite Pro Mechanical BIM

19.4.1 Description

Opens the **Netload** dialog box to select a DLL file and load the application.

19.5 NEW command

Opens the **Select Template** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

19.5.1 Description

Opens the **Select Template** dialog box to select a DWT or DWG file to use as a template for creating a new drawing.

19.6 NEWSHEETSET command

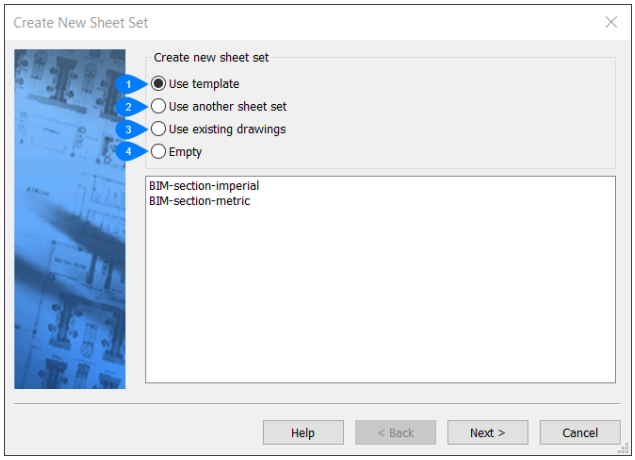
Opens the **Create New Sheet Set** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

19.6.1 Description

Opens the **Create New Sheet Set** dialog box to create a new sheet set.



1 Use template

2 Use another sheet set

3 Use existing drawings

4 Empty

19.6.2 Use template

Use template

Creates 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 *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.

1 ***Note:*** All metric projects use by default a sheet set template in mm, and will honor the setting of the SECTIONSHEETSETTEMPLATEMETRIC system variable, if not empty. The cm and meter sheet set and sheet template files are no longer included in the installation.

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.

Note: By default, the current drawing's location folder is open.

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, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set Manager** panel with the new sheet set.

19.6.3 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 the **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 for the sheet set and save it.

Important: You cannot progress until you have selected 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, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

19.6.4 Use existing drawings

Creates 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.

Note: The dialog box fills with names of drawings and their layouts.

2 Choose 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. This 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 for the sheet set and save it.

Important: 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, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

19.6.5 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 for the sheet set and save it.

Important: 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, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

19.7 NEWWIZ command

Opens the **Create New Drawing** dialog box.

Shape Lite Pro Mechanical BIM

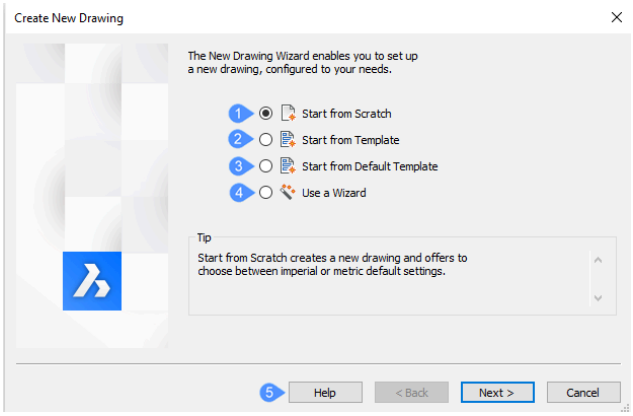
Icon:

Alias: DDNEW

19.7.1 Description

Opens the **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

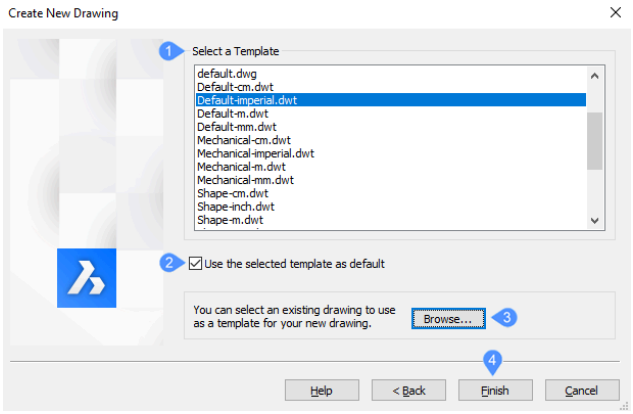
19.7.2 Start from Scratch

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.

19.7.3 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\%username%\AppData\Local\Bricsys\BricsCAD\V25x64\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 system 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.

19.7.4 Start from Default Template

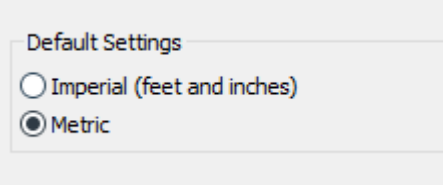
Starts new drawings with the default template defined by the BASEFILE system variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

19.7.5 Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way:

1 Click the **Next** button.

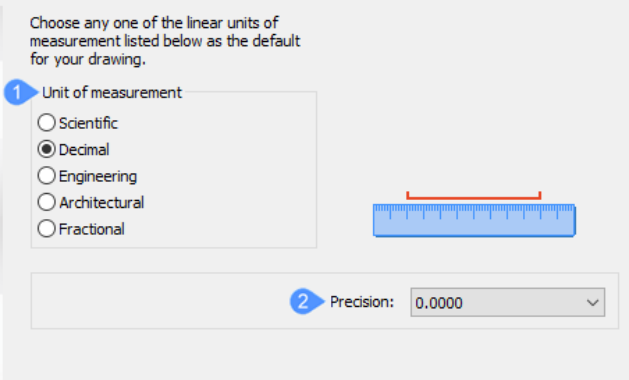
2 Choose a measurement system:



- 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.



- Unit of Measurement
- 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.

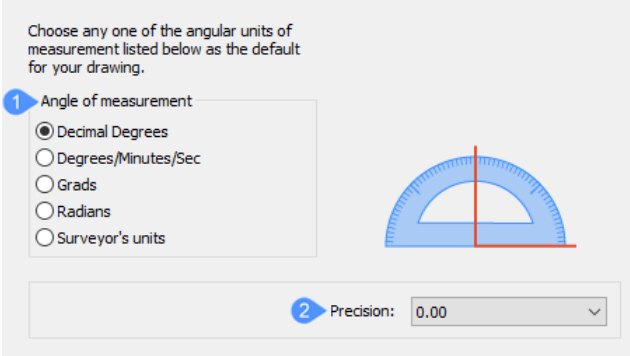
Note: This is a display precision: the program makes calculations internally always to eight decimal places.

Note:

1 Click the **Next** button.

2 Select an angular unit of measurement and display precision.

3



- Angle of measurement
- Precision

Angle of Measurement

Choose one from the list. If you are unsure, choose Decimal/Degrees; you can always change the units any time later with the UNITS command.

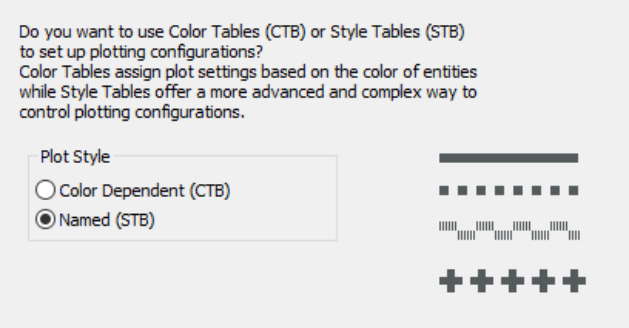
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds	Degrees,minutes,seconds,decimal secs	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units	North/South degrees towards East/West	N12d34'56"E

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:

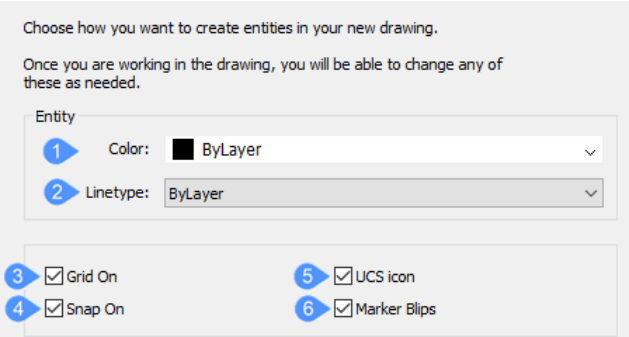
3 

- **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.

4 Click the **Next** button.

5 Choose the default working color, the linetype and the visual aids you want to be displayed by the drawing:

6 

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.

19.7.6 Help

Opens the Bricsys Help article about the NEWWIZ command.

19.8 NODE command

Toggles the **Node** entity snap.

Shape Lite Pro Mechanical BIM

Icon:

19.8.1 Description

Toggles the **Node** entity snap to enable or disable snapping to node. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

19.9 NONE command

Disables all 2D entity snaps.

Shape Lite Pro Mechanical BIM

Icon: 

19.9.1 Description

Disables all 2D entity snaps to prevent the cursor from snapping to 2D entities. You can launch this command at the Command prompt to turn off running entity snaps. Doing so changes the value of the OSMODE system variable to zero. You can also launch this command within another command to turn off entity snaps only for the current operation. This does not change the value of the OSMODE system variable.

This command displays no prompts and has no options.

19.10 NUMBER command

Creates incremental number tags for BIM entities.

Shape | Lite | Pro | Mechanical | BIM

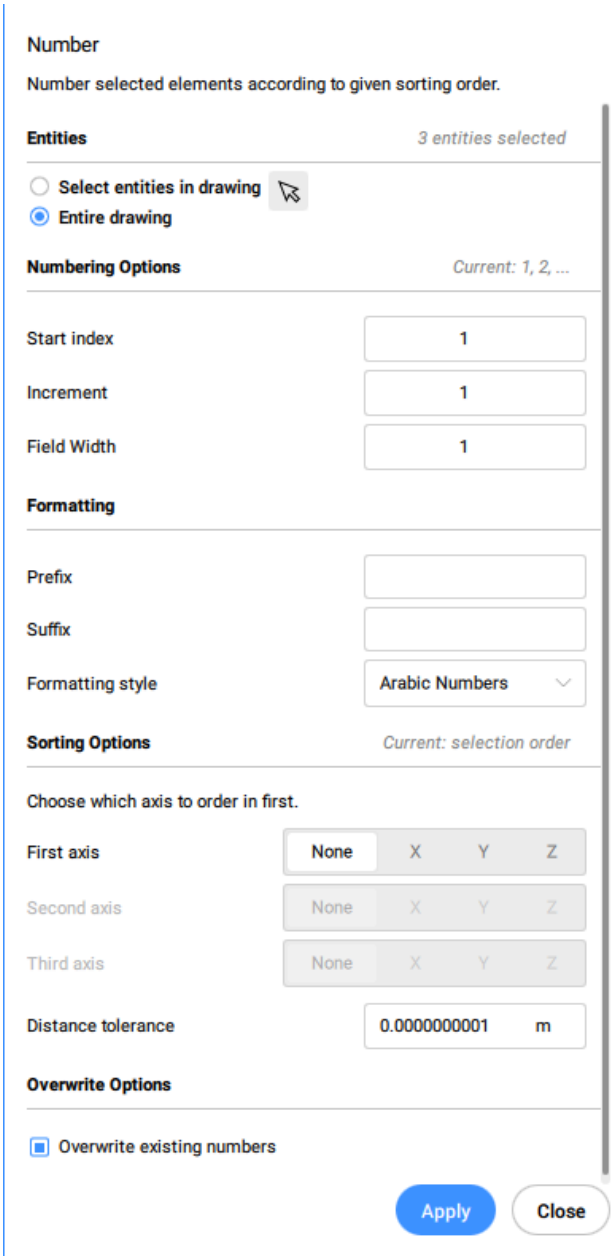
Icon:

19.10.1 Description

Provides the first index for the selected entities.

Note: Integer numbers are accepted.

The command opens the Number command panel.



Select entities:

- **Select entities in the drawing**: choose which entities to select.
- **Entire drawing**: by default, the panel selects all BIM entities in the model space.

Numbering Options

- **Start index:** sets the index from which to start the numbering.
- **Increment**: sets the increment used for numbering
- **Field Width**: sets the total length of the number field. This will add the appropriate amount of leading zeros in front of Arabic numbers, so that the length of the resulting string is equal to the field width.

Formatting

- **Prefix**: sets a prefix for the numbering.
- **Suffix**: sets a suffix for the numbering.
- **Formatting style**: choose a number style from the drop-down list.

Sorting Options

Allows to specify which axis to order first.

X

Based on the value of the x-coordinate, from low to high.

Y

Based on the value of the y-coordinate, from low to high.

Z

Based on the value of the z-coordinate, from low to high.

None

Uses selection order, unless the option 'Entire Drawing' was chosen, in which case it will order from oldest to most recent.

Distance Tolerance

Sets the distance tolerance for sorting in comparison. Two solid's centroids within a given tolerance are considered equal, and they will be numbered in selection order.

Overwrite Options

Allows to overwrite or to keep existing numbers.

19.10.2 Options within the command line

Increment

Sets the increment for the numbering.

Prefix

Specifies a prefix for the numbering.

Suffix

Specifies a suffix for the numbering.

Number style

Defines the number style.

0
Define Arabic numbers (1, 2, 3, ...).

1
Define roman uppercase numbers (I, II, III, ...).

2
Define roman lowercase numbers (i, ii, iii, ...).

3
Define uppercase letters (A, B, C, ...).

4
Define lowercase letters (a, b, c, ...).

Field Width
Sets the total length of the number field. This will add the appropriate amount of leading zeros in front of Arabic numbers.

Entities sorting
Allows to specify the sorting order.

X
Based on the value of the x-coordinate, from low to high.

Y
Based on the value of the y-coordinate, from low to high.

Z
Based on the value of the z-coordinate, from low to high.

None
Uses selection order, unless the option **Entire Drawing** was selected, in which case it will order from oldest to most recent.

Tolerance
Sets the distance tolerance for sorting in comparison. Two solid's centroids within a given tolerance are considered equal, and they will be numbered in selection order.

Overwrite Numbers
Allows to overwrite or to keep existing numbers.

Kept
Keeps existing numbers.

Overwritten
Overwrites existing numbers.

Change Selection
Allows to change the selection set.

20. O

20.1 OBJECTSCALE command

Adds or removes scale factors.

Shape Lite Pro Mechanical BIM

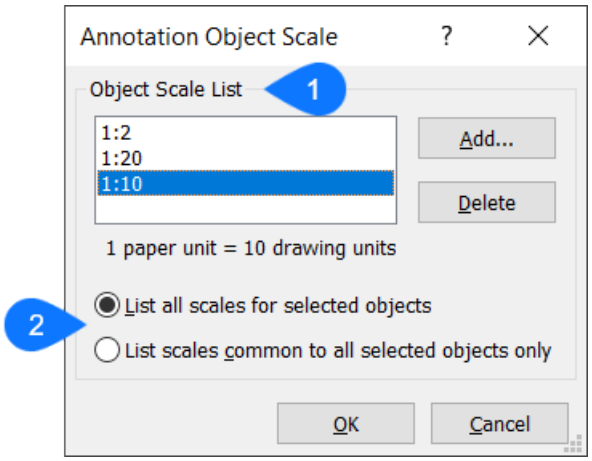
Icon:

20.1.1 Description

Adds or removes scale factors used with annotative entities through **Annotation Object Scale** dialog box.

Note: Annotative scaling adjust scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

The **Annotation Object Scale** dialog box allows you to add or remove scale factors used with annotative entities.



1 Object Scale List

2 Options

20.1.2 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.

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

20.1.3 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.

Note: At least two entities must be selected.

20.2 -OBJECTSCALE command

Adds or removes scale factors.

Shape Lite Pro Mechanical BIM

20.2.1 Description

Adds or removes scale factors used with annotative entities through **Annotation Object Scale** dialog box.

Note: Annotative scaling adjust scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

20.2.2 Options within the command

Select annotative entities
Select one or more items using any selection method.

Note: Write all in the Command line to select all entities in the drawing. The program filters out ineligible entities automatically.

Add
Adds annotation scales to the selected annotative entities.

Note: Type **?** to list the annotation scale factors available.

Delete
Deletes annotation scales from the selected annotative entities.

?
Displays a list of available annotation scales as defines in the scale list.

Note: See SCALELISTEDIT command.

	Scale Name	Paper Units	Drawing Units	Effective Scale
1:	100:1	100.0000	1.0000	100.0000
2:	10:1	10.0000	1.0000	10.0000
3:	1:1	1.0000	1.0000	1.0000
4:	1:10	1.0000	10.0000	0.1000
5:	1:100	1.0000	100.0000	0.0100
6:	1:125	1.0000	125.0000	0.0080
7:	1:16	1.0000	16.0000	0.0625
8:	1:2	1.0000	2.0000	0.5000
9:	1:20	1.0000	20.0000	0.0500
10:	1:25	1.0000	25.0000	0.0400
11:	1:250	1.0000	250.0000	0.0040
12:	1:30	1.0000	30.0000	0.0333
13:	1:4	1.0000	4.0000	0.2500
14:	1:40	1.0000	40.0000	0.0250
15:	1:5	1.0000	5.0000	0.2000
16:	1:50	1.0000	50.0000	0.0200
17:	1:8	1.0000	8.0000	0.1250
18:	2:1	2.0000	1.0000	2.0000
19:	4:1	4.0000	1.0000	4.0000
20:	8:1	8.0000	1.0000	8.0000

20.3 OFFSET command

Creates parallel copies of lines, polylines, splines and edges.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: O

20.3.1 Description

Creates copies of linear entities and align them parallel to the original entities at a specified distance.

Note:

- The SELECTIONMODES system variable controls what is selected by default (whole entities, subentities or boundaries) to avoid potential conflicts.
- The OFFSETGAPTYPE system variable controls how possible gaps, in parallel copies of closed polylines, are filled.
- When curved entities are offset, the command changes the radius of the copies.

20.3.2 Method

Enter the offset distance and select the side for parallel copy.

20.3.3 Options within the command

Enter offset distance
Specifies the distance between the original entity and its parallel copy.

Note: The offset distance is saved to the OFFSETDIST variable.

Both Sides
Places the copies on both sides of the entity.

Through point
Specifies the offset distance by picking a point in the drawing area.

Multiple
Allows you to make multiple copies using the original entity.

next entity/subentity
Allows you to select a new entity or subentity to create multiple copies.

Erase
Toggles whether to delete the source entity after the offset operation or not.

Note: The status is saved to the OFFSETERASE variable with the drawing, but is then reset (turned Off) with the next drawing.

Layer
Specifies the layer on which to place the copied entity.

Current
Places the offset entity on the current layer.

Source
Places the offset entity on the same layer as the source entity.

Undo
Undoes the last offset operation.

Exit
Exits the command.

20.4 OFFSET3D command

Creates a copy of an entity at a specified offset distance and elevation difference from the original.

Shape Lite Pro Mechanical BIM

Icon:

20.4.1 Description

Creates copies of lines, polylines, 3D polylines, or Civil Strings, and aligns them parallel to the original entity at a specified offset distance and elevation difference.

Note: The SELECTIONMODES system variable controls what is selected by default (whole entities, subentities or boundaries) to avoid potential conflicts.

20.4.2 Method

Enter the offset distance, then the elevation difference and select the side for parallel copy.

20.4.3 Options within the command

Enter offset distance
Specifies the distance between the original entity and its parallel copy.

Both Sides
Places the offset copies on both sides of the entity.

Through point
Specifies the offset distance by picking a point in the drawing area.

Grade
Allows you to specify a grade in %.

Slope
Allows you to specify a slope (1:x).

Elevation
Specifies the elevation difference between the original entity and its parallel copy.

Erase
Toggles whether to delete the source entity after the offset operation or not.

Layer
Specifies the layer on which to place the copied entity.

Current
Places the offset entity on the current layer.

Source
Places the offset entity on the same layer as the source entity.

Exit
Exits the command.

20.5 OLELINKS command

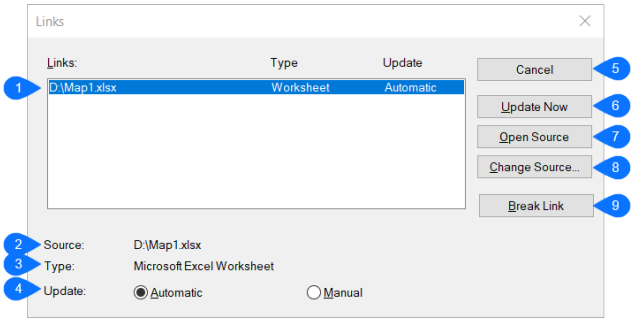
Opens the **Links** dialog box.

Shape Lite Pro Mechanical BIM

20.5.1 Description

Opens the **Links** dialog box to view and manage OLE links.

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

20.5.2 Links

Provides a list of all links loaded in the file.

20.5.3 Source

Reports the path to the source file name.

20.5.4 Type

Reports the file type.

20.5.5 Update

Determines how the OLE entity in the drawing is updated when the source changes.

20.5.6 Cancel

Cancels the linking operation.

20.5.7 Update Now

Updates linked OLE entities.

20.5.8 Open Source

Opens the source file in the source application. This allows you to edit the source file.

20.5.9 Change Source

Changes the source for the linked OLE entity.

20.5.10 Break Link

Breaks the link with the source file.

Note: When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.

20.6 OLEOPEN command

Opens the selected OLE object.

Shape Lite Pro Mechanical BIM

20.6.1 Description

Opens the selected OLE object in the source application for editing. (short for "object linking and embedding linking"). If the link between the drawing and the source application is broken, the OLE object cannot open.

Note: Use the INSERTOBJ command to insert OLE objects in drawings.

Note: The command is only available on the windows platform.

20.7 ONWEB command

Opens the default Web browser.

Shape Lite Pro Mechanical BIM

20.7.1 Description

Opens the default Web browser to browse the internet starting from the BricsCAD home page. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

20.8 OOPS command

Restores the last erased entity.

Shape Lite Pro Mechanical BIM

Icon:



Alias: UNDELETE, UNERASE

20.8.1 Description

Restores the last erased entity, including those erased by the Block command. If there are no erased entities in the drawing, BricsCAD indicates: There is nothing to un-erase.

This command displays no prompts and has no options.

20.9 OPEN command

Opens the **Open** file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

20.9.1 Description

Opens the **Open** file dialog box to select a file to open in the drawing editor.

Supported open file formats are:

- **Standard Drawing File** (.dwg)

  Warning: The minimum supported version of AutoCAD® Mechanical drawings is AutoCAD® Mechanical 2012, and the maximum is AutoCAD® Mechanical 2021. If you try to open an ACM drawing created with an unsupported version, a warning notification message appears: "**This file was created in an unsupported Mechanical DWG drawing version. It is recommended to save it first as Mechanical DWG 2021 drawing then open it again in BricsCAD. Do you want to open it as read-only?**".

  You can choose from the following options:

  - **Yes**: Opens the drawing in read-only mode and allows you to view and possibly edit the drawing without saving it.
  - **No**: Stops opening of a drawing and returns you to the previous state/screen.
- **Drawing Exchange Format** (.dxf)

- **Template Format** (.dwt)
- **Standards File** (.dws)
- **Windows Metafile Formats** (.wmf; .emf; .wmz; .emz)(1) (2)
- **Collada** (.dae)
- **Micro Station DGN File** (.dgn)

  Formats available with the BIM add-on:
- **Rhino file** (.3dm)(1) (2)
- **SketchUp file**(.skp)(1) (2)
- **IFC file** (.ifc; .ifczip)
- **Revit Family** (.rfa)(1) (2)
- **Revit Project** (.rvt)(1) (2)

  (1) Not available in BricsCAD for Linux.

  (2) Not available in BricsCAD for macOS.

20.10 OPENSHEETSET command

Opens the **Select a Sheet Set file** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

20.10.1 Description

Opens the **Select a Sheet Set file** dialog box to select a DST file to open in the **Sheet Sets** panel.

20.11 -OPENSHEETSET command

Opens a sheet set file.

Shape Lite Pro Mechanical BIM

20.11.1 Description

Opens a DST sheet set file, and then displays the **Sheet Sets** panel which lists all drawings and sheets associated with the set; meant for use by macros. An example of path and file name of a DST sheetset file is *F:\work\Documentation\Documentation\project.dst*.

20.12 OPTIMIZE command

Corrects inaccuracies in the drawing, for 2D entities or 3D entities.

Shape Lite Pro Mechanical BIM

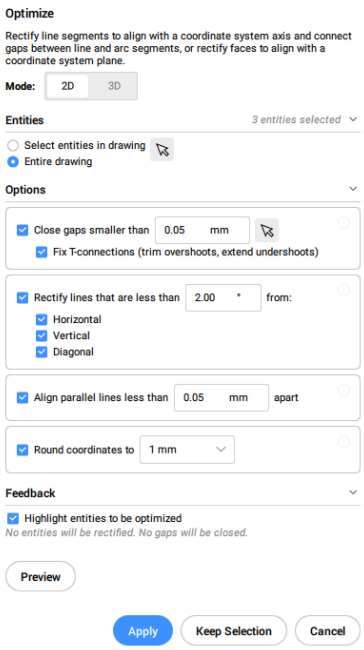
Icon:

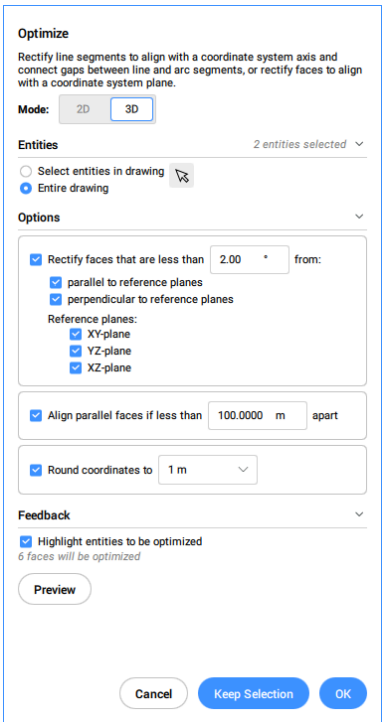
20.12.1 Description

This command works with either 2D entities or 3D entities. The supported 2D entities are lines, arcs and polylines. The command makes corrections such as small gaps between lines or near vertical, horizontal and diagonal lines.

The supported 3D entities are solids, 3D regions and surfaces. The command makes corrections such as faces that are near parallel to the orthogonal planes to be coplanar to these planes; and faces that belong to different solids to be coplanar to one another.

This command opens the **Optimize** command context panel.





Note:

- The options within the OPTIMIZE command are the same as in the **Optimize** command context panel.
- The **Optimize** command context panel allows you to toggle On/Off the live Feedback. When turned On, it highlights the entities that match the selected options and displays the number of entities that will be optimized.

20.12.2 Methods

There are two methods to begin optimizing entities:

Entire drawing
When this option is selected, all non-frozen entities in the current viewport are used as input.

selection options (?)
Allows to choose a selection method. See the SELECT command.

20.12.3 Options within the command, in 2D mode

change reference Angles
This option determines which entities will be optimized based on their orientation.

Note: In the **Optimize** command context panel, set the value for the **Rectify lines that are less than** option.

Horizontal lines
Corrects near horizontal lines according to the angle tolerance.

Vertical lines
Corrects near vertical lines according to the angle tolerance.

45-Degree lines
Corrects near 45° lines according to the angle tolerance.

All
Corrects near horizontal, near vertical and near 45° lines according to the angle tolerance.

None
Does not correct entities based on their orientation.

change Tolerances
Allows you to set the tolerances for the parameters used during the optimization.

Angle tolerance
Sets the angle tolerance in degrees, with respect to the World Coordinate System (WCS). Near horizontal, vertical or diagonal lines within this angle tolerance will be optimized.

Note: In the **Optimize** command context panel, set the value for the **Rectify lines that are less than** option.

Distance tolerance
Sets the distance tolerance (see **Note**). Parallel lines within this distance will be merged.

Note: In the **Optimize** command context panel, set the value for the **Align parallel lines less than** option.

Rounding precision
Sets the rounding precision (see **Note**). This option rounds the coordinates to the specified number of digits following the decimal.

Note: In the **Optimize** command context panel set, the value for the **Round coordinates to** option.

Gap tolerance
Sets the gap tolerance (see **Note**). Gaps between collinear lines within the gap tolerance will be filled.

Note: In the **Optimize** command context panel, set the value for the **Close gaps smaller than** option. Gaps smaller than this value will be closed with a transparent red circles in the drawing. You can also use the picker () to set the gap tolerance by specifying two points in the drawing.

When the Fix T-connections (trim overshoots, extend undershoots) option is check, the lines are trimmed and extended to close T-connections for smaller gaps than gap tolerance value.

Note: The values respects the insertion units of the drawing (see the INSUNITS system variable).

change Options
Determines if gaps will be closed, if the lines will be aligned with a referenced direction, and if the drawing will be rounded.

close Gaps
If this option is enabled, the gaps between collinear lines that are smaller than the gap tolerance will be closed.

Note: In the **Optimize** command context panel tick the **Close gaps smaller than** checkbox.

rectify Lines
If this option is enabled, entities will be aligned along the referenced directions (horizontal, vertical or diagonal) when their deviation are less than the specified angle tolerance.

Note: In the **Optimize** command context panel tick the **Rectify lines that are less than** checkbox.

Note: If this option is disabled, the **Round Coordinates** option will be disabled too.

Make lines Collinear
If this option is enabled, lines that are almost collinear will be aligned as collinear.

Note: In the **Optimize** command context panel tick the **Align parallel lines less than** checkbox.

Round coordinates
If this option is enabled, it rounds the coordinates to the rounding precision.
If this option is disabled, the drawing will not be rounded.

Note: In the **Optimize** command context panel tick the **Round coordinates to** checkbox.

Note: This action is controlled by the **Rounding precision** tolerance setting.

enable All
Enables all **close Gaps**, **Make lines Collinear**, and **Round coordinates** options.

change Input selection
Allows you to make a new selection of entities to be used by the command.

apply settings and PReview
Makes a preview of the rectified segments. You can accept or reset the result.

Keep selection
Clicking this button will cancel the command without optimizing, but with keeping the highlighted lines selected. This option allows you to select suboptimal lines in a drawing.

switch to 3d Mode
Switches the mode to 3D.

20.12.4 Options within the command, in 3D mode

change reference Plane
Determines which plane will be the referenced for correcting faces position. You can choose XY plane, YZ plane, XZ plane or all of them.

change Tolerances
This option allows setting the tolerances for the parameters used during the optimization.

Angle tolerance
Sets the angle tolerance, in degrees, for a relative position of faces and planes.

Note: In the **Optimize** command context panel, set the value for the **Rectify faces that are less than** option.

Distance tolerance
Sets the distance tolerance (see **Note**).
The faces within the tolerance will be made coplanar, parallel or perpendicular, according to the settings.

Note: In the **Optimize** command context panel, set the value for the **Align parallel faces if less than** option.

Rounding precision
Sets the rounding precision (see **Note**). This option rounds the coordinates to the specified number of digits following the decimal.

Note: In the **Optimize** command context panel, set the value for the **Round coordinates to** option.

Note: The values respects the insertion units of the drawing (see the INSUNITS system variable).

change Options
Changes the options for correcting the faces position.

make faces Coplanar
Makes the faces in the selection coplanar, with respect to the values in the **Tolerances** section.

rectify faces PArallel to reference planes
Makes the faces in the selection parallel to the selected reference planes, with respect to the values in the **Tolerances** section.

rectify faces PErpendicular to reference planes
Makes the faces in the selection perpendicular to the selected reference planes, with respect to the values in the **Tolerances** section.

Round coordinates
If this option is enabled, it rounds the coordinates to the rounding precision.
If this option is disabled, the drawing will not be rounded.

Note: This action is controlled by the **Rounding precision** setting.

enable All
All the options for 3D faces (coplanar, parallel, perpendicular and round) will be used by the command.

change Input selection
Allows making a new selection of entities to be used by the command.

apply settings and PReview
Makes a preview of the rectified faces. You can accept or reset the result.

Keep selection
Clicking this button will cancel the command without optimizing, but with keeping the highlighted entities selected. This option allows you to select suboptimal entities in a drawing.

switch to 2d Mode
Switches the mode to 2D.

20.13 OPTIONS command

Opens the **Settings** dialog box with the **Program options** category expanded.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: CFG, CONFIG, OP, PREFERENCES, PREFS

20.13.1 Description

Opens the **Settings** dialog box with the **Program options** category expanded to view and modify relevant system variables.

20.14 ORTHOGONAL command

Toggles the ORTHOMODE system variable.

Shape Lite Pro Mechanical BIM

Alias: OR, ORTHO

20.14.1 Description

Toggles the ORTHOMODE system variable to specify if the cursor is constrained to dragging at 90 degree angles. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'ORTHOGONAL.

- On: turns on the ORTHOMODE system variable.
- Off: turns off the ORTHOMODE system variable.
- Toggle: changes the ORTHOMODE system variable to the opposite of the current setting.

20.15 OSNAP command

Opens the **Settings** dialog box with the **Entity snap mode** category expanded.

Shape Lite Pro Mechanical BIM

Alias: DDESNAP, DDOSNAP, OS, SETESNAP

20.15.1 Description

Opens the **Settings** dialog box with the **Entity snap mode** category expanded to view and modify relevant system variables.

20.16 -OSNAP command

Toggles entity snap modes.

Shape Lite Pro Mechanical BIM

Alias: -OS, ESNAP

20.16.1 Description

Toggles entity snap modes at the Command line (short for "object snap").

Note: Entity snaps help you draw and edit precisely by snapping the cursor to the nearest geometric feature.

20.16.2 Options within the command

NEArest

Toggles **Nearest** entity snap mode. Snaps to the nearest point on any entity.

Note: Can also toggle with the NEAREST command.

ENDpoint

Toggles **Endpoint** entity snap mode. Snaps to the endpoints of open entities, such as lines, arcs, open polylines, and open splines.

Note: Can also be toggled with the ENDPOINT command.

MIDpoint
Toggles **Midpoint** entity snap mode. Snaps to the midpoint of open entities.

Note: Can also be toggled with the MIDPOINT command.

CENter
Toggles **Center** entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs.

Note: Can also be toggled with the CENTER command.

GeometriC cEnter
Toggles **Geometric center** entity snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.

Note: Can also be toggled with the GCENTER command.

PERpendicular
Toggles **Perpendicular** entity snap mode. Snaps to the perpendicular of one entity to another.

Note: Can also be toggled with the PERPENDICULAR command.

TANgent
Toggles **Tangent** entity snap mode. Snaps to the tangencies of circular entities.

Note: Can also be toggled with the TANGENT command.

QUAdrant
Toggles **Quadrant** entity snap mode. Snaps to the quadrant points of circular entities, such as circles, arcs, and polyarcs.

Note: Can also be toggled with the QUADRANT command.

INSertion point
Toggles **Insertion** entity snap mode. Snaps to the insertion point of blocks and text.

Note: Can also be toggled with the INSERTION command.

POInt
Toggles **Node** entity snap mode. Snaps to point entities.

Note: Can also be toggled with the NODE command.

EXTension
Toggles **Extension** entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection.

Note: Can also be toggled with the EXTENSION command.

PARallel
Toggles **Parallel** entity snap mode. Snaps to the parallel point of entities. Allows to draw lines, polyline segments, xlines and rays parallel to another entity.

Note: Can also be toggled with the PARALLEL command.

INTersection
Toggles the **Intersection** entity snap mode. Snaps to the intersection of any pair of entities.

Note: Can also be toggled with the INTERSECTION and 3DINTERSECTION commands.

APParent intersection
Toggles the **Apparent intersection** entity snap mode. Snaps to the apparent intersection of entities in 3D space.

Note: Can also be toggled with the APPARENT and 2DINTERSECTION commands.

clear entity snaps (NON)
Clears entity snaps.

ON
Turns all entity snap modes on.

OFF
Turns all entity snap modes off.

Note: Modes can also be turned off with the NONE command.

20.17 OVERKILL command

Deletes duplicate and overlapping entities and combines duplicate blocks.

Shape Lite Pro Mechanical BIM

Icon: 

20.17.1 Description

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 and optionally purges duplicate block definitions.

After selecting the entities, the **Delete Duplicate Entities** dialog box displays. For more information, see the related article **Delete Duplicate Entities dialog box**.

20.17.2 Options within the command

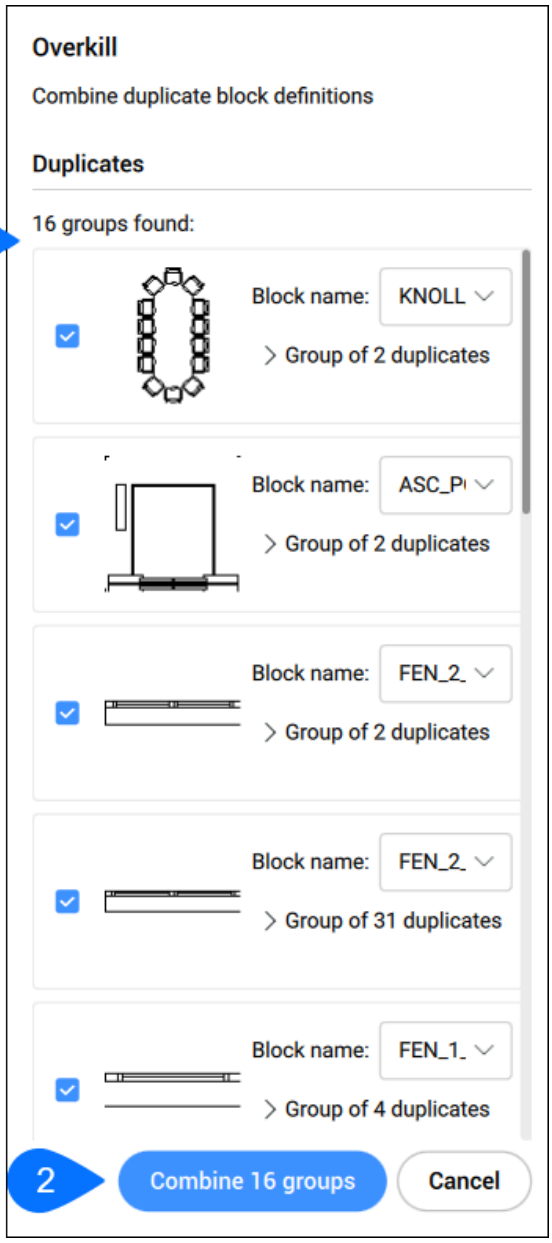
Instead of selecting entities, another option can also be selected.

Combine duplicate block definitions
This options displays the **Combine Duplicate Block Definition** dialog box. Here you can define entity properties to ignore and set tolerances. For more information, see the related article **Combine Duplicate Block Definition dialog box**.

Once all preferences are defined and applied, the **Overkill** Command Context panel displays.

20.17.3 Overkill command context panel

The **Overkill** Command Context panel allows you to specify which duplicate block definitions should be combined.



1 **Block list**

Displays a list of all groups of duplicate block definitions found. Choose which groups to combine by ticking the corresponding selection boxes.

For each group is displayed the number and the names of duplicate block definitions found. Expand a group and click on any block definition name to highlight its corresponding instances in the drawing.

2 **Combine**

Combines all selected duplicate block definitions of the list.

20.18 -OVERKILL command

Deletes duplicate and overlapping entities and combines duplicate blocks.

Shape Lite Pro Mechanical BIM

20.18.1 Description

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 and optionally purges duplicate block definitions.

20.18.2 Options within the command

Done
Executes the overkill procedure.

Ignore
Sets the properties to ignore. You can choose between **None**, **All**, **Color**, **LAyer**, **Ltype**, **ltScale**, **LWeight**, **Thickness**, **TRansparency**, **plotSTyle**, and **Material**.

None
Properties are taken into account to determine if overlapping entities should be combined.

All
All properties are taken into account.

Color, etc.
The colors of overlapping entities are considered. If the colors match, then they are combined or deleted.

tOlerance
Sets the tolerance for the comparison process.

Note: When is set to 0, entities must match completely before being evaluated in the overkill process.

optimize Plines
Segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. If lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline.

segment witDth
Specifies if the width property of polyline segments is ignored.

Break polyline
Polyline segments are kept intact even if the duplicates are deleted.

Yes
Optimizes polyline entities.

No
Polyline entities are not optimized.

combine parTial overlap
Partially overlapping entities are unified into a single entity.

combine Endtoend
Collinear lines and polyline segments having coinciding endpoints are unified into a single entity or segment.

Associativity
Associative entities are not processed.

ignore Solids
Specifies if duplicate 3D solids are ignored or removed.

delete or Move duplicates
Determines what to do with duplicate entities.

Delete
Deletes duplicate entities.

Move
Entities are moved to Overkill Layer.

Note: This layer is set by the OVERKILLLAYER system variable. By default, it is set to **Duplicate Entities**.

Combine duplicate block definitions
Searches the drawing for duplicate block definitions and replaces all such block instances by the most recent of these duplicates.

Purge duplicate block definitions
Deletes duplicate block definitions.

21. P

21.1 PACKAGE command

Creates a package ZIP file of the drawing and its dependencies.

Shape Lite Pro Mechanical BIM

See the -ETRANSMIT command.

21.2 PAGESETUP command

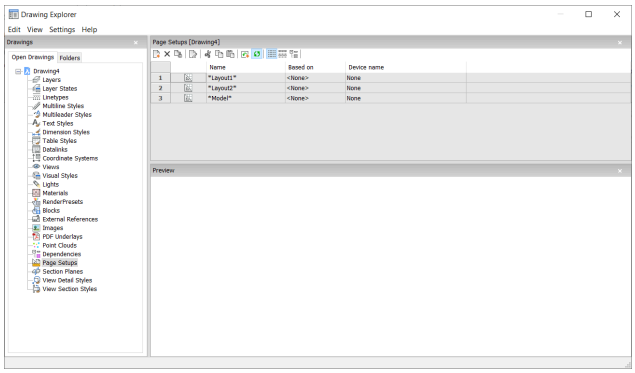
Opens the **Drawing Explorer** dialog box with **Page Setups** selected.

Shape Lite Pro Mechanical BIM

Icon:

21.2.1 Description

Opens the **Drawing Explorer** dialog box with **Page Setups** category selected to view and modify page setups in the current drawing.



21.2.2 Context Menu Options

New

Loads additional page setup definitions into the drawing. Displays the **New Page Setup** dialog box (see **New Page Setup dialog box** related article).

Delete

Deletes page setup definitions from the drawing. The following page setup definitions cannot be deleted:

- The Model tab
- The last layout

Edit Page Setup

Edit the selected page setup. Displays the **Page Setup** dialog box. Here you can edit the properties of the selected page setup (see **Page Setup dialog box** related article).

Rename

Renames the selected page setup.

Note: The following page setups cannot be renamed:

- The Model tab
- The Layout tabs

Select All
Selects all page setup definitions.

Invert selection
Deselects the current selection and vice versa.

21.3 PAN command

Moves the entire drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: P

21.3.1 Description

Moves the entire drawing in real- time within the current viewport.

Note: Real-time means that the drawing moves as the mouse moves.

Note: The PERSPECTIVE system variable must be set to 0.

After running the command, the cursor changes into a hand icon. Hold down the left mouse key and then move the mouse to pan the drawing or right-click to open the Pan context menu.

Note: Pan the drawing also with the scroll bars. Turn them on with the SCROLLBAR command.

21.4 -PAN command

Moves the entire drawing.

Shape Lite Pro Mechanical BIM

Alias: -P

21.4.1 Description

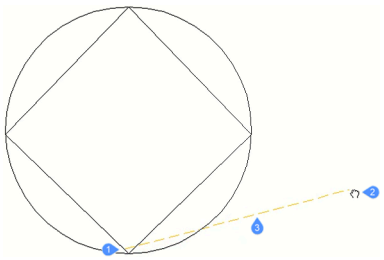
Moves the entire drawing within the current viewport.

Note: The PERSPECTIVE system variable must be set to 0.

21.4.2 Method

There are two methods to use -PAN command:

- Pick a base point and a displacement point.
- Choose one of the predefined options.



1 Pan base point.

2 Pan cursor which specifies the pan displacement point.

3 Pan displacement.

21.4.3 Options within the command

Pan base point
Specifies the pan distance's starting point.

Pan displacement point
Specifies the pan distance's ending point.

Note: The drawing is panned by 5% or 100% of the width of the current viewport.

Left
Pans the drawing 5% to the right.

Right
Pans the drawing 5% to the left.

Up
Pans the drawing 5% downwards.

Down
Pans the drawing 5% upwards.

PaGe Left
Pans the drawing 100% to the right.

Note: You can also press Shift + Left cursor key.

PaGe Right
Pans the drawing 100% to the left.

Note: You can also press Shift + Right cursor key.

PaGe Up
Pans the drawing 100% upwards.

Note: You can also press Shift + Up cursor key.

PaGe Down
Pans the drawing 100% downwards.

Note: You can also press Shift + Down cursor key.

21.5 PANELIZE command

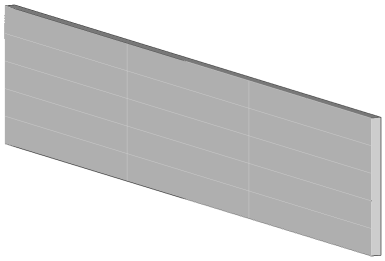
Creates a grid on the face of a 3D Solid, as a block reference or mesh.

Shape Lite Pro Mechanical BIM

Icon:

21.5.1 Description

Creates a custom grid on the face of a 3D Solid, by mentioning the length and the number of panels.



21.5.2 Options within the command

Length U panels
Sets the length of the panels in U direction (height).

Length V panels
Sets the length of the panels in V direction (width).

Number U panels
Number of panels in U direction (rows).

Number V panels
Number of panels in V direction (columns).

Result
Toggles between **Polylines** and **Mesh**.

Polylines
Creates the grid as a block reference.

Mesh
Creates the grid as a Mesh entity.

21.6 PARALLEL command

Toggles the **Parallel** entity snap.

Shape Lite Pro Mechanical BIM

Icon:

21.6.1 Description

Toggles the **Parallel** entity snap to enable or disable snapping to parallel. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

21.7 -PARAMETERS command

Manages dimensional-constraint parameters.

Shape Lite Pro Mechanical BIM

21.7.1 Description

Creates, edits, renames, and deletes parametric equations made of dimensional constraints and user-defined variables; works at the Command line.

21.7.2 Options within the command

?
Lists all dimensional constraints and user variables in the current drawing, showing the name, expression and current value of each dimensional constraint or user variable.

New
Creates a user variable.

Edit
Edits the expression for the dimensional constraint or user variable.

Rename
Renames a dimensional constraint or user variable.

Delete
Deletes a dimensional constraint or user variable.

Properties
Sets the lower and the upper bounds for the parameter.

Lower bound
Defines the lower value for the parameter.

Upper bound
Defines the upper value for the parameter.

Note: The upper and lower bounds are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

21.8 PARAMETERSPANELCLOSE command

Closes the **Parameters Manager** panel.

Shape Lite Pro Mechanical BIM

21.8.1 Description

Closes the **Parameters Manager** panel to hide it from the current workspace. If the **Parameters Manager** panel is stacked when you close it, the **Parameters Manager** panel tab or icon is removed from the stack.

21.9 PARAMETERSPANELOPEN command

Opens the **Parameters Manager** panel.

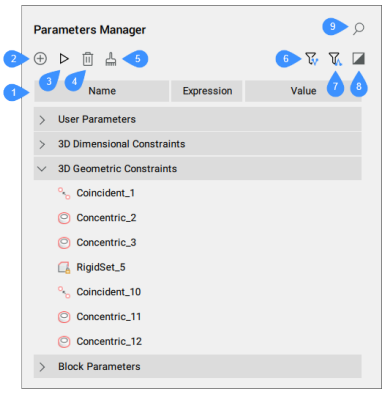
Shape Lite Pro Mechanical BIM

Icon:

21.9.1 Description

Opens the **Parameters Manager** panel to display it in the current workspace. The **Parameters Manager** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Parameters Manager** panel can be either floating, docked or stacked.

The **Parameters Manager** panel is the main tool to view and modify parameters and constraints in the current drawing.



1 Parameters list

2 New parameter

3 Animate

4 Delete

5 Clean unused variables

6 Show dependent parameters

7 Show defining parameters

8 Show selected

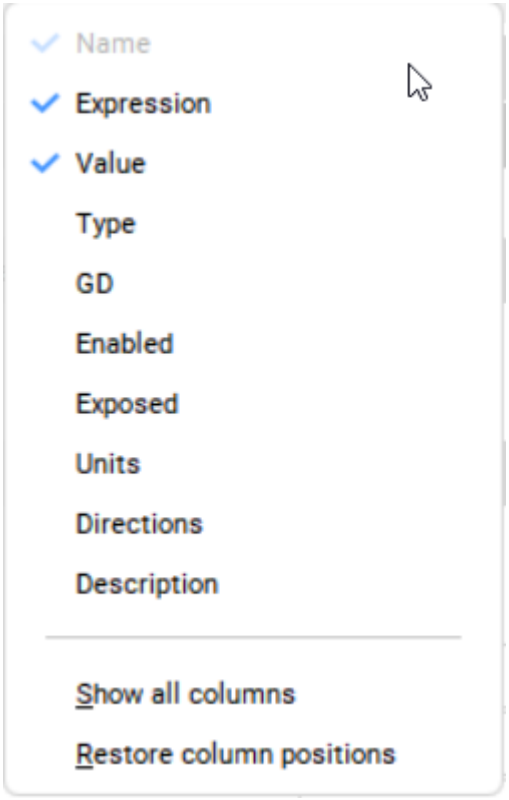
9 Search

21.9.2 Parameter list

Displays a list of all the parameters in the drawing. The parameters are organized in expandable categories which can include user parameters (including the ones which are linked with Design Tables), 2D

dimensional constraints, 3D dimensional constraints, reference parameters, 2D geometric constraints, 3D geometric constraints, block (components) parameters, and array parameters.

The parameter list header contains columns which represent the properties of the parameters. Columns can be moved to a new position, resized or hidden. You can arrange parameters in ascending or descending order by pressing a specific column head. The parameter list header has three columns by default: **Name**, **Expression** and **Value**. To add more columns, right click the columns header and select the desired columns from the list:



Note: Some properties are read-only (for example **Type**), others can be modified by the user (for example **Name**).

Name

Displays the name of the parameter.

Note: The name can be edited. When you rename an item, all instances of the item are updated in the drawing and in the **Parameters Manager** parameters list.

Expression

Specifies the mathematical expression of the parameter. You can edit the expression by clicking on the cell.

Value

Displays the value resulting from the parameter's expression. The value is a read-only real number, unless the parameter is linked to a Design table. In that case, the parameter has a list of possible values represented as a combo-box. You can change the parameter's value by picking a new value from the list. The

combo-box is searchable by typing a sub-string of the value to quickly find it. This is useful for large lists of values.

Type
Displays the parameter's type:

- Constraint type for constraints (Distance, Concentric etc.).
- Parameter's type for user and block parameters (Real, String).
- Parameter's type for array parameters (Real, Boolean, Enumeration).

GD (geometry-driven)
If ON, the parameter is geometry-driven.

Lower bound
Defines the lower value for the parameter.

Upper bound
Defines the upper value for the parameter.

Note: The upper and lower bounds are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

Enabled
Defines whether a geometric constraint parameter is taken into consideration.

Exposed
Defines whether the parameter is visible and can be modified when the component is inserted in an assembly.

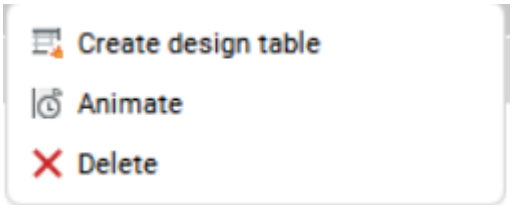
Units

Sets the dimension of the parameter. You can choose between: none, linear (mm), square (mm^2) and cubic (mm^3).

Directions
Defines the direction for a geometric constraint parameter (any, same, opposite or reset).

Description
Allows you to type a description for the user parameters.

Right-clicking on a selected parameter opens a specific context-menu that contains only those options, which apply to that item. These options can also be found in the context menu for the same item in the **Mechanical Browser** panel.



Create design table
Creates a design table to drive parametric block parameters.

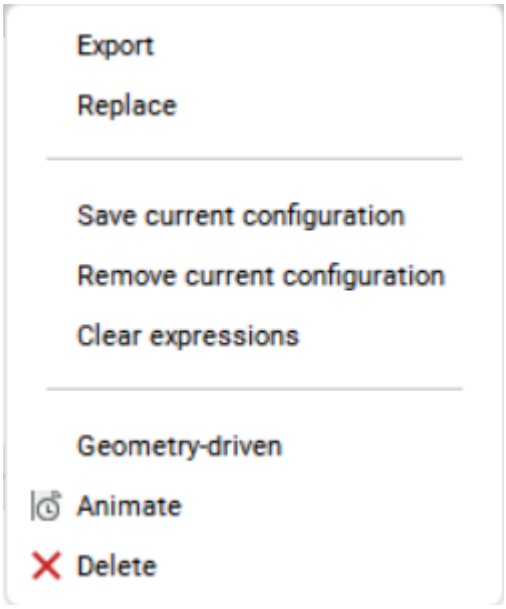
Animate
Animates the parameter value within some range.

Link to parameter
Links the subcomponent parameter to the main level parameter.



Unlink
Breaks the links between the subcomponent parameter and the main level parameter.

Geometry-driven
If ticked, the parameter is geometry-driven.



Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries will be taken into account while moving the constrained geometry.

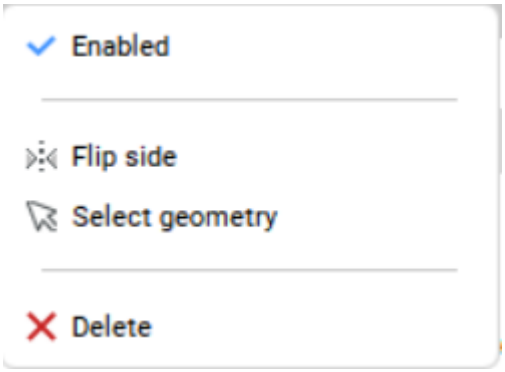
Export
Opens the **Select design table file** dialog box to save the design table.

Replace
Opens the **Select design table file** dialog box to choose a design table to replace the existing one.

Save current configuration
Saves the current design table configuration.

Remove current configuration
Removes the current design table configuration.

Clear expression
Removes the expression formula.



Enabled
Displays and toggles the enabled status of the constraint.

Flip side
Flips the side of the constraint.

Select geometry
Selects the entities involved in the selected constraint in the model space.

21.9.3 New parameter

Inserts a new user parameter with default values for **Expression** and **Value**.

21.9.4 Animate

Animates the selected parameter value within a range.

21.9.5 Delete

Removes selected parameters and constraints from the drawing.

21.9.6 Clean unused variables

Erases unused variables.

21.9.7 Show dependent parameters

When enabled, the panel shows parameters and constraints, depending on the selected parameter(s).

21.9.8 Show defining parameters

When enabled, the panel shows parameters which define the selected parameter(s).

21.9.9 Show selected

When enabled, the panel shows parameters and constraints for the objects currently selected in the model space, or all objects if nothing is selected.

21.9.10 Search

Filters the parameter list to display only the parameter names that match the character string you enter in the search box.

Note: You can remove the active filter by pressing the Close (X) button from the right side of the search box.

21.10 PARAMETRICBLOCKIFY command

Automatically replaces sets of entities, matching a selected parametric block or a set of parametrized entities in a model, by parametric block references.

Shape Lite Pro Mechanical BIM

Icon: 

21.10.1 Description

Searches the drawing for identical sets of entities (2D or 3D) and replaces them with block references.

21.10.2 Method

There are four methods to use the BLOCKIFY command:

- Detect matches with selected entities
- Detect matches with existing blocks
- Detect equal 3D solids
- Detect collections (of 3D solids/block references and 2D block references)

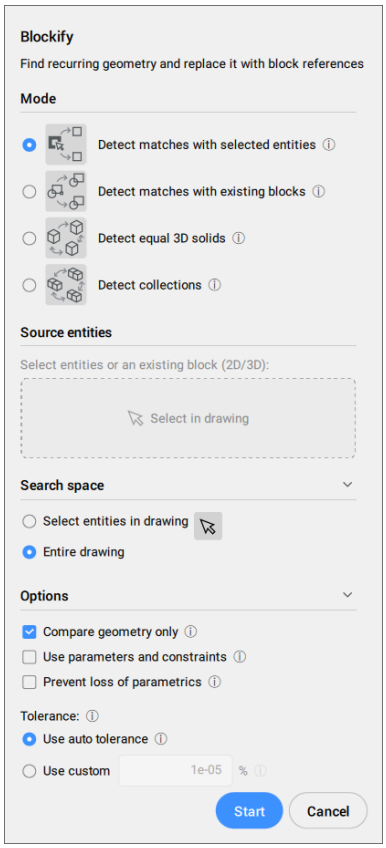
Dependent on the selected mode, different follow-up questions and advanced options are presented.

Note: To replace block patterns by arrays of entities, use the ARRAYDETECT command. See the **ARRAYDETECT command** related article.

Select the desired method, source entities and search space. Set the options and tolerance as needed. The options displayed are relevant for the selected method and entities and reflect the BLOCKIFYMODE and BLOCKIFYTOLERANCE system variables settings (see the related articles).

21.10.3 Detect matches with selected entities

Searches the drawing (or the selection) for recurring 2D/3D geometry and converts them to block references.



Source entities

Click **Select in drawing** button. Select entities or an existing block (2D/3D) in the drawing to compare other entities to.

Search space

Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing

Uses the entire drawing as search space.

Options

Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Use parameters and constraints

Try to find matching geometry by varying parameter values.

Note:

- When the option is active, a selection set containing constraints between possible matches (external constraints) is not considered valid. In this case, the Command line displays a warning prompt to remove these constraints before using the command.
- Potentially slower on large files.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

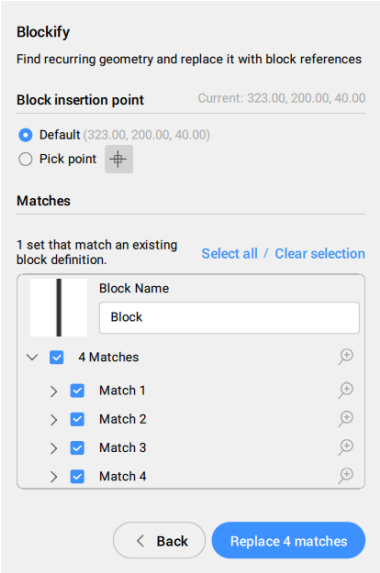
This option is highly recommended for most cases.

Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



Block insertion point

Define the insertion point of the new block definition.

Default

Accept the default insertion point marked by a red X in the model space.

Pick point
Allows you to define a new insertion point. Press the icon next to the option and select a point in the model space.

Matches
Displays a list of detected matches and allows you to select the ones to replace with block references:

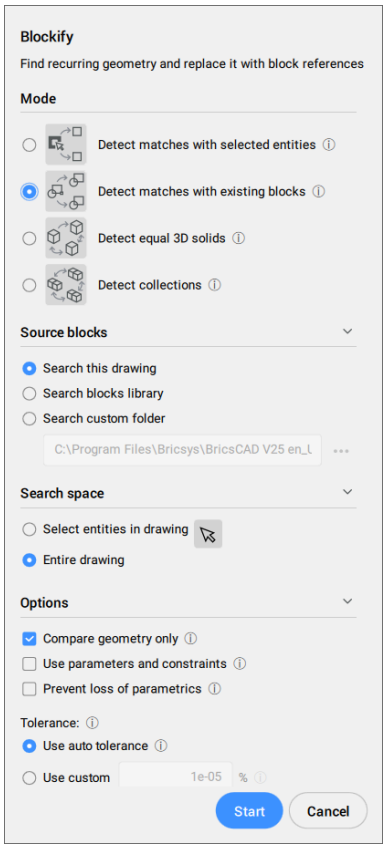
- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.
- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

21.10.4 Detect matches with existing blocks

Searches the drawing (or the selection) for matches to existing blocks from a specified location and converts them to block references.



Source blocks
Allows you to specify the source blocks search location.

Search this drawing
Search for source blocks in the current drawing.

Search blocks library
Search for source blocks in the blocks library.

Search custom file/folder
Check this radio button to activate the input field. Click the menu button to open the **Select Folder** dialog box and indicate the custom file/folder.

Search space
Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing
Uses the entire drawing as search space.

Options
Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only
Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Use parameters and constraints
Try to find matching geometry by varying parameter values.

Note:

- When the option is active, a selection set containing constraints between possible matches (external constraints) is not considered valid. In this case, the Command line displays a warning prompt to remove these constraints before using the command.
- Potentially slower on large files.

Prevent loss of parametrics
If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

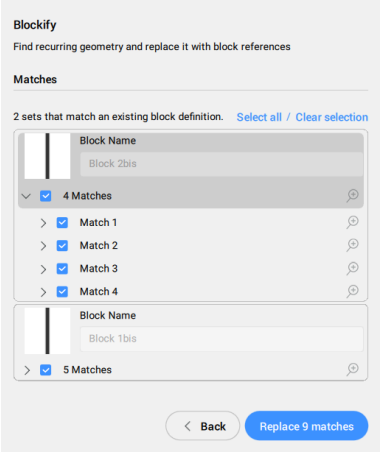
Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance
Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.
For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance
Check this radio button to use an automatically set value.
The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).
This option is highly recommended for most cases.

Use custom
Check this radio button to activate the input field.
The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



Matches

Displays a list of detected matches and allows you to select the ones to replace with block references:

- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

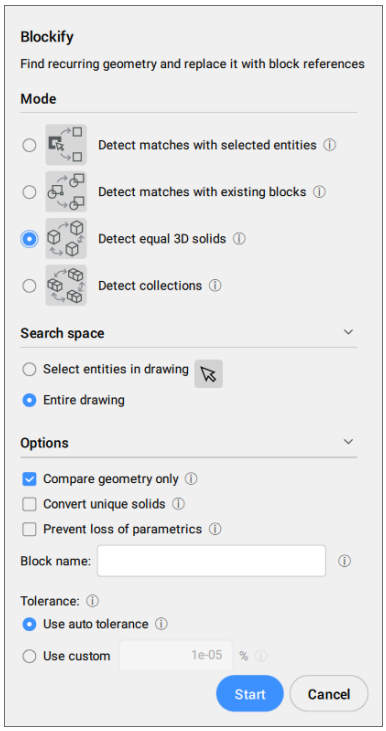
Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.
- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

21.10.5 Detect equal 3D solids

Searches the drawing (or the selection) for equally shaped solids and converts them to block references (new block definitions). In case a solid matches an existing block definition, the replacing block reference points to that block definition.

Note: This option is not available in the BricsCAD Lite license level.



Search space

Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing

Uses the entire drawing as search space.

Options

Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Convert unique solids

Tick the check-box to also convert solids to blocks when they occur only once.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Block name

Specify a block name or use the default name. If you choose to use the default name, block definitions with names like **Block1**, **Block2** etc. are created.

> ***Note:*** If BLOCKIFY is used on BIM entities, it checks the BIM Name property of all equal-shaped BIM classified solids and tries to find the longest common substring to use it as the name of the block definitions.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

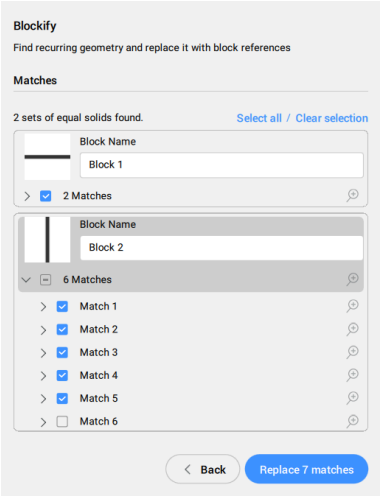
This option is highly recommended for most cases.

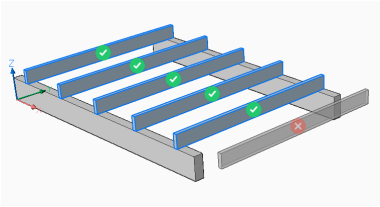
Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.





Matches

Displays a list of detected matches and allows you to select the ones to replace with block references:

- Tick the group check-boxes in the panel.
- Click the individual check marks / x signs in the drawing.

Note:

- To visually identify matches/sets from the list in the model space, click their name in the list. To add or remove matches/sets to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.

- To zoom in the view to a set/match, click the magnifying glass symbol next to its name.

21.10.6 Detect collections

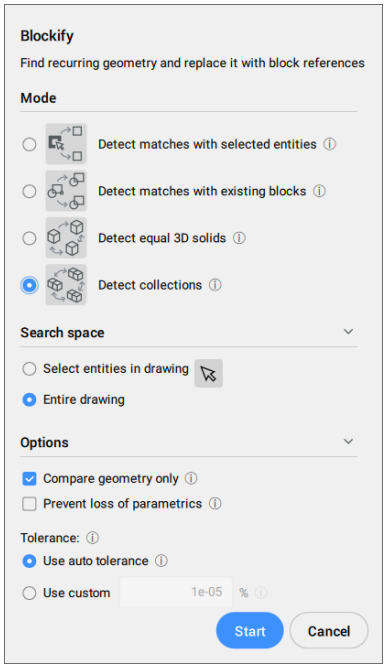
Searches the drawing (or the selection) for sets of identical collections of:

- 3D solids and/or 3D block references
- 2D block references

Allows you to select one set at a time from which to create a new block definition. Replaces the collections within the selected set with block references of the new definition.

Recalculates the collection sets taking into account the newly created block reference (some previously found collection sets configurations might no longer be available).

Note: With a large number of input entities, calculating all possible collections can take a long time. Press the **Esc** key to cancel the calculation at any time, and then select from the collection sets found so far.



Search space
Define how the entities to be compared with the source entities are selected.

Select entities in the drawing

Allows you to manually select entities by displaying the selection pick box. Click the **Select** arrow ( ) to create a new selection set.

Entire drawing
Uses the entire drawing as search space.

Options
Options that modify the current value of the BLOCKIFYMODE system variable:

Compare geometry only

Ignores color, layer, line type, scale, weight and thickness, transparency, plot style and material properties when comparing entities.

Prevent loss of parametrics

If set, no constraints or parametric operations will be removed. Parameters and constraints are preserved and remapped to the subentities of the replacing blocks.

Options that modify the current value of the BLOCKIFYTOLERANCE system variable (specifies the relative tolerance for determining whether two entities are equal):

Tolerance

Defines the tolerance as percentage for entity comparisons, specifying the allowable variation in sizes or positions.

For example, two lines with lengths of 10 and 9 would be considered equal with a tolerance of 10%.

Use auto tolerance

Check this radio button to use an automatically set value.

The algorithm will automatically select the best tolerance (0.0001% for 2D and 0.03% for 3D).

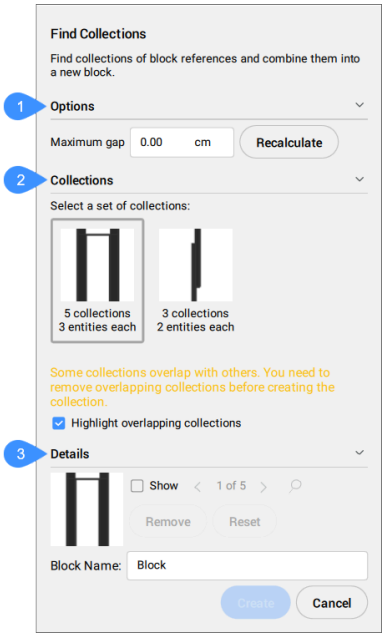
This option is highly recommended for most cases.

Use custom

Check this radio button to activate the input field.

The custom tolerance should be between 0 and 100 percentage.

Note: The tolerance value is relative. Unexpected behavior might happen for values more than 5%.



1 Options

2 Collections

3 Details

Options

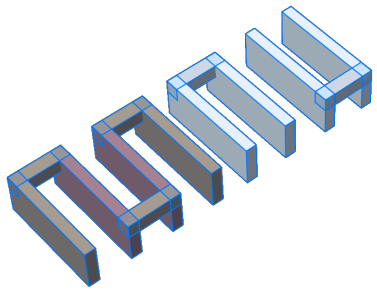
Maximum gap
Increases the allowed gap between entities to find a more complex collection.
For example: If a drawing contains multiple block references of desks and chairs, you can find a collection of both elements by setting the maximum gap between a desk and a chair.

Recalculate
Recalculates the collections based on the new maximum gap.

Collections
Displays the found collection sets. Select the collection set you want to create the block reference from.

Note:

- A message is displayed if the selected set contains overlapping collections. Overlapping collections are collections that have at least one entity in common. Select which overlapping collections to remove so you can create the new block definition.
- The overlapping collections are highlighted if the **Highlight overlapping collections** option is ticked.



Details

Show
Tick the check-box to highlight in the **Model space** one collection at a time within the collection set.
To cycle through the collections, use the left and right arrows next to the **Show** check-box.
Use the **Zoom in** button to extend the view to the highlighted collection.

Remove
Removes the current highlighted collection from the collection set.

Reset
Resets to the initially found collections in the collection set by re-adding any collections that were removed.

21.11 PARAMETRICMOVE command

Defines a parameter to be used by a move operation on entities of a parametric block.

Shape Lite Pro Mechanical BIM

Icon:

Alias: PMOVE

21.11.1 Description

This command is used with 2D parametric blocks and allows you to move entities in a block reference.

21.11.2 Method

Specify the base point and the second point of displacement, then select the entities which will be affected by the move operation.

Note: The command works similarly to the PARAMETRICSTRETCH command, whose workflow is described in the article **2D parametric blocks**.

Note: You can define parametric move, stretch, and rotate operations in non-XY planes after changing the UCS.

21.11.3 Options within the command

Base point of displacement
Allows you to select a location point for the origin of the move vector.

Second point of displacement
Allows you to select a location point for the head of the move vector.

Select entities which can be affected by the operation
Allows you to edit the selection of entities to be affected by the move operation. All selected entities will be highlighted.

change Linked behavior
Allows you to decide whether the selected entities will follow operation definition points when adjusted by other operations. The **OFf** option corresponding to the **not linked** behavior is set as default for newly created parametric operations. Select **ON** to set the behavior as **linked**.

Note: When affected by a parametric stretch operation, linked parametric move operations behave differently from their unlinked variant.

Enter name for the action parameter
Allows you to give a name to the move parameter. The default name is **Move**.
The block entities that were selected will have their position adjusted according to the new value of the move parameter.

The appearance of the block can be controlled by changing the value of the parameter associated with the operation:

- Select the parametric block and drag the parameter's grip point (hover the cursor over the grip point to display the current value of the parameter).
- Access this parameter in the **Parameters Manager** panel / **Mechanical Browser** panel.

Note: Use the PARAMETRICMOVEEDIT command to edit a move operation.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.

- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

21.12 PARAMETRICMOVEEDIT command

Edits parametric move operations.

Shape Lite Pro Mechanical BIM

Alias: PMOVEEDIT

21.12.1 Method

The command lets you edit an existing move operation.

Launch the command, enter the name of the desired move operation, then select the parametric operation data to be edited.

Note: To dynamically edit a move operation within a parametric block, select the block and drag the operation's grip point.

21.12.2 Options within the command

Shownames
Lists the names of all the move operations available in the drawing.

Selection
Lets you edit the elements selection of the specified operation.

Direction
Lets you define a new operation direction by selecting its base and second points of displacement.

Linked behavior
Allows you to decide whether the entities will follow operation definition points when adjusted by other operations. The **not linked** behavior is set as default for newly created parametric operations.

Note: The **Linked** property can also be turned **On** or **Off** from the bottom section of the **Mechanical Browser**. The **On** value is signaled in **Parameters Manager** and **Mechanical Browser** panels by the symbol ∞.

Note: When affected by a parametric stretch operation, linked parametric move operations behave differently from their unlinked variant.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

21.13 PARAMETRICOFFSET command (Experimental)

Defines a parameter to be used by an offset operation on curves of a parametric block.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: POFFSET

21.13.1 Disclaimer

EXP.

Parametric Offset is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

21.13.2 Description

This command is used with 2D parametric blocks and allows you to offset planar curves in a block reference.

21.13.3 Method

Select one or more planar curves and specify an offset distance.

Note: The command works similarly to the PARAMETRICSTRETCH command, whose workflow is described in the article **2D parametric blocks**.

21.13.4 Options within the command

Select curves
Selects the planar curves to be affected by the offset operation.

Offset distance
Specifies the distance between the original entity and its parallel copy.

Through point
Specifies the offset distance by picking a point in the drawing area.

Enter name for the operation parameter
Allows you to give a name to the offset parameter. The default name is **Offset**.

The appearance of the block can be controlled by changing the value of the parameter associated with the operation:

- Select the parametric block and drag the parameter's grip point (hover the cursor over the grip point to display the current value of the parameter).
- Access this parameter in the **Parameters Manager** panel / **Mechanical Browser** panel.

Note: Use the PARAMETRICOFFSETEDIT command to edit an offset operation.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.

- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

21.14 PARAMETRICOFFSETEDIT command (Experimental)

Edits an existing offset operation.

Shape Lite Pro Mechanical BIM

Alias: POFFSETEDIT

21.14.1 Disclaimer

EXP.

Parametric Offset is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

21.14.2 Method

Enter the name of the desired offset operation, then select the parametric operation data to be edited.

Note: To dynamically edit an offset operation within a parametric block, select the block and drag the operation's grip point.

21.14.3 Options within the command

Shownames
Lists the names of all the offset operations available in the drawing.

selection options (?)
Lets you change the selection set of the specified operation.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

21.15 PARAMETRICROTATE command

Defines a parameter to be used by a rotate operation on entities of a parametric block.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PROTATE

21.15.1 Method

This command is used with 2D parametric blocks.

Allows you to rotate entities in a block.

Specify the rotation center point, rotation arc start point, rotation arc end point, then select the entities which will be affected by the rotate operation.

Note: The command works similarly to the PARAMETRICSTRETCH command, whose workflow is described in the article **2D parametric blocks**.

Note: You can define parametric move, stretch, and rotate operations in non-XY planes after changing the UCS.

21.15.2 Options within the command

Select rotation center point
Allows you to select a location point for the origin of the rotate vector.

Select rotation arc start point
Allows you to select a location point for the head of the rotation arc.

Select rotation arc end point (hold CTRL to switch direction)
Allows you to select a location point for the end of the rotation arc.

Select/deselect entities which can be affected by the operation
Allows you to edit the selection of entities to be affected by the rotate operation. All selected entities will be highlighted.

Enter name for the operation parameter
Allows you to give a name to the rotate parameter. The default name is **Rotate**.
The block entities that were selected will have their position adjusted according to the new value of the rotate parameter.

change Linked behavior
Allows you to decide whether the selected entities will follow operation definition points when adjusted by other operations. The **OFf** option corresponding to the **not linked** behavior is set as default for newly created parametric operations. Select **ON** to set the behavior as **linked**.

The appearance of the block can be controlled by changing the value of the parameter associated with the operation:

- Select the parametric block and drag the parameter's grip point (hover the cursor over the grip point to display the current value of the parameter).
- Access this parameter in the **Parameters Manager** panel / **Mechanical Browser** panel.

Note: Use the PARAMETRICROTATEEDIT command to edit a rotate operation.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.
- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

21.16 PARAMETRICROTATEEDIT command

Edits parametric rotate operations.

Shape Lite Pro Mechanical BIM

Alias: PROTATEEDIT

21.16.1 Method

The command lets you edit an existing rotate operation.

Enter the name of the desired rotate operation, then select the parametric operation data to be edited.

Note: To dynamically edit a rotate operation within a parametric block, select the block and drag the operation's grip point.

21.16.2 Options within the command

Shownames
Lists the names of all the rotate operations available in the drawing.

Selection
Lets you edit the elements selection of the specified operation.

Geometry
Lets you define a new operation geometry by selecting a rotation center point and rotation arc start and end points.

Linked behavior
Allows you to decide whether the entities will follow operation definition points when adjusted by other operations. The **not linked** behavior is set as default for newly created parametric operations.

Note: The **Linked** property can also be turned **On** or **Off** from the bottom section of the **Mechanical Browser**. The **On** value is signaled in **Parameters Manager** and **Mechanical Browser** panels by the symbol ∞.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

21.17 PARAMETRICSCALE command

Defines a parameter to be used by a scale operation on entities of a parametric block.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PSCALE

21.17.1 Method

Allows you to scale entities in a block.

Select the base point of displacement, the second point of displacement and the entities which can be affected by the operation.

Note: The command works similarly to the PARAMETRICSTRETCH command, whose workflow is described in the article **2D parametric blocks**.

21.17.2 Options within the command

Base point of displacement
Allows you to select a location point for the origin of the scale vector.

Second point of displacement
Allows you to select a location point for the head of the scale vector.

Select/deselect entities which can be affected by the operation
Allows you to edit the selection of entities to be affected by the scale operation. All selected entities will be highlighted.

change Linked behavior
Allows you to decide whether the selected entities will follow operation definition points when adjusted by other operations. The **OFf** option corresponding to the **not linked** behavior is set as default for newly created parametric operations. Select **ON** to set the behavior as **linked**.

Note: Scale operations can affect and can be affected by any type of other operations.

Enter name for the operation parameter
Allows you to give a name to the scale parameter. The default name is **Scale**.
The block entities that were selected will have their dimension adjusted according to the new value of the scale parameter.

The appearance of the block can be controlled by changing the value of the parameter associated with the operation:

- Select the parametric block and drag the parameter's grip point (hover the cursor over the grip point to display the current value of the parameter).
- Access this parameter in the **Parameters Manager** panel / **Mechanical Browser** panel.

Note: Use the PARAMETRICSCALEEDIT command to edit a scale operation.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.
- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

21.18 PARAMETRICSCALEEDIT command

Edits parametric scale operations.

Shape Lite Pro Mechanical BIM

Alias: PSCALEEDIT

21.18.1 Method

The command lets you edit the scale frame, elements selection, direction and linked behavior of an existing scale operation.

Enter the name of the desired scale operation, then select the parametric operation data to be edited.

Note: To dynamically edit a scale operation within a parametric block, select the block and drag the operation's grip point.

21.18.2 Options within the command

Shownames
Lists the names of all the scale operations available in the drawing.

Selection
Lets you edit the scale frame and elements selection of the specified operation.

Direction
Lets you define a new operation direction by selecting its base and second points of displacement.

Linked behavior
Allows you to decide whether the entities will follow operation definition points when adjusted by other operations. The **not linked** behavior is set as default for newly created parametric operations.

Note: The **Linked** property can also be turned **On** or **Off** from the bottom section of the **Mechanical Browser**. The **On** value is signaled in **Parameters Manager** and **Mechanical Browser** panels by the symbol ∞.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

21.19 PARAMETRICSTRETCH command

Defines a parameter to be used by a stretch operation on some entities of a parametric block.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PSTRETCH

21.19.1 Description

The goal of this parametric stretch operation is to simplify the stretching of entities in the block. Usually, the same behavior could be obtained with parameters and constraints, but it would take more time and effort to obtain the same behavior.

21.19.2 Method

Allows you to stretch some entities in a block.

Specify the base point and the second point of displacement, set the stretch frame, then select the entities which will be affected by the stretch operation.

Note: You can define parametric move, stretch, and rotate operations in non-XY planes after changing the UCS.

Note: The stretch parameter might not work well in combination with the geometric and dimensional constraints. The behavior of the block when applying the stretch parameter together with constraints is highly dependent on the complexity of the geometry and the number and type of the constraints.

The appearance of the block can be controlled by changing the value of the parameter associated with the operation:

- Select the parametric block and drag the parameter's grip point (hover the cursor over the grip point to display the current value of the parameter).
- Access this parameter in the **Parameters Manager** panel / **Mechanical Browser** panel.

For more info about using the PARAMETRICSTRETCH command, see the article **2D parametric blocks**.

21.19.3 Options within the command

Base point of displacement
Allows you to select a location point for the origin of the stretch vector.

Second point of displacement
Allows you to select a location point for the head of the stretch vector.

Construct stretch frame: pick first point of rectangular frame
Allows you to select the first corner of the rectangular stretch frame.

Opposite corner
Allows you to select the second corner of the rectangular stretch frame.

Polygonal
Allows you to enter a series of points that will set a polygonal stretch frame.

Note: The vertices inside the selection contour (rectangle or polygon) will be moved according to the stretch vector.

Select/deselect entities which can be affected by the operation
Allows you to edit the selection of entities to be affected by the stretch operation. All the selected entities will be highlighted.
By default, all the entities which have stretch points inside the stretch frame are selected.

change Linked behavior
Allows you to decide whether the selected entities will follow operation definition points when adjusted by other operations. The **OFf** option corresponding to the **not linked** behavior is set as default for newly created parametric operations. Select **ON** to set the behavior as **linked**.

Enter name for the action parameter
Allows you to give a name to the stretch parameter. The default name is **Stretch**.
The block entities that were selected for stretch will adjust according to the new value of the stretch parameter.

21.20 PARAMETRICSTRETCHEDIT command

Edits parametric stretch operations.

Shape Lite Pro Mechanical BIM

Alias: PSTRETCHEDIT

21.20.1 Method

The command lets you edit an existing stretch operation.

Enter the name of the desired stretch operation, then select the parametric operation data to be edited.

Note: To dynamically edit a stretch operation within a parametric block, select the block and drag the stretch operation's grip point.

21.20.2 Options within the command

Shownames
Lists the names of all the stretch operations available in the drawing.

Selection
Lets you edit the stretch frame and elements selection of the specified operation.

Direction
Lets you define a new operation direction by selecting its base and second points of displacement.

Linked behavior
Allows you to decide whether the entities will follow operation definition points when adjusted by other operations. The **not linked** behavior is set as default for newly created parametric operations.

Note: The **Linked** property can also be turned **On** or **Off** from the bottom section of the **Mechanical Browser**. The **On** value is signaled in **Parameters Manager** and **Mechanical Browser** panels by the symbol ∞.

Note: All editing options can also be accessed from the operation's right-click context menu in the **Parameters Manager** and **Mechanical Browser** panels.

21.21 PARAMETRIZE command

Applies constraints automatically.

Shape Lite Pro Mechanical BIM

21.21.1 Description

Automatically adds geometric constraints and parametric equations to 3D solid geometry. The resulting parameters and geometric constraints can be viewed and edited in the **Mechanical Browser** panel (MECHANICALBROWSEROPEN command).

When the ROLLOVERPARAMS system variable is ON, the parameters can also be edited from Model Space via Rollover Tips when hovering over the blocks.

Note: To activate the Rollover Tips (RT) option, set the ROLLOVERTIPS system variable to ON or activate the **RT** button in the **Status bar**.

21.22 PARAMETRIZE2D command

Automatically applies 2D geometric and dimensional constraints to selected 2D geometry.

Shape Lite Pro Mechanical BIM

21.22.1 Description

Select the 2D entities to automatically add dimensional and geometric constraints.

The parameters and constraints can be edited in the **Properties** panel, the **Parameters Manager** panel, or the **Mechanical Browser** panel.

When the ROLLOVERPARAMS system variable is ON, the parameters can also be edited from Model Space via Rollover Tips when hovering over the blocks.

Note: To activate the Rollover Tips (RT) option, set the ROLLOVERTIPS system variable to ON or activate the **RT** button in the **Status bar**.

21.23 PASTEBLOCK command

Pastes CAD entities from the Clipboard as a block.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can use this command, first use the COPYCLIP or COPYBASE commands to copy entities from the current drawing, or even from another CAD program.

21.23.1 Description

This command can only paste CAD entities into the drawing as block entities. Other Clipboard content may be pasted as OLE entities.

21.24 PASTECLIP command

Pastes CAD entities from the Clipboard into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can use this command, first use the COPYCLIP or COPYBASE commands to copy entities from the current drawing, or even from another CAD program.

21.24.1 Method

This command can only paste CAD entities into the drawing as entities.

When the Clipboard contains non-BricsCAD entities:

- Non-BricsCAD entities, such as raster images, are pasted as OLE entities.
- If non-BricsCAD text (including LISP and Diesel code) is pasted in the Command line, the program executes the text as commands.

21.24.2 Options within the command

Rotate
Specifies the angle by which to rotate the entities.

Scale
Specifies the scale factor.

Mirror
Mirrors the entities about a mirror line in a 2D plane.

21.25 PASTEORIG command

Pastes entities from the clipboard into a different drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can use this command, first use the COPYCLIP or COPYBASE command to copy entities from the current drawing.

Note: The PASTEORIG command works concomitantly with the INSUNITS system variable. The INSUNITS system variable must have the same value in both source and destination drawing.

21.25.1 Description

Pastes CAD entities from the Clipboard into a different drawing using the coordinates of the entities in the source drawing. PASTEORIG comes from **paste at original coordinates**.

Note: This command does not paste entities back into the source drawing.

21.26 PASTESPEC command

Opens the **Paste Special** dialog box.

Shape Lite Pro Mechanical BIM

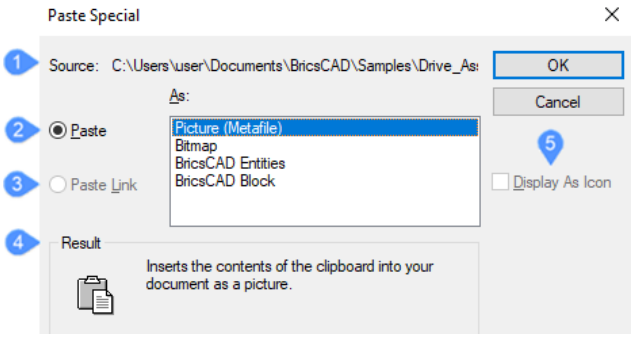
Icon: 

Alias: PA

Note: This is a Windows only command.

21.26.1 Description

Opens the **Paste Special** dialog box to 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

21.26.2 Source

Displays the path where the drawing/entity/block can be found.

21.26.3 Paste As

Chooses the format in which to paste the entities. This list varies, depending on the content of the Clipboard. When the Clipboard contains data copied from a BricsCAD drawing, then the following formats are available:

- **Picture Metafile**: entities are converted to 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:*

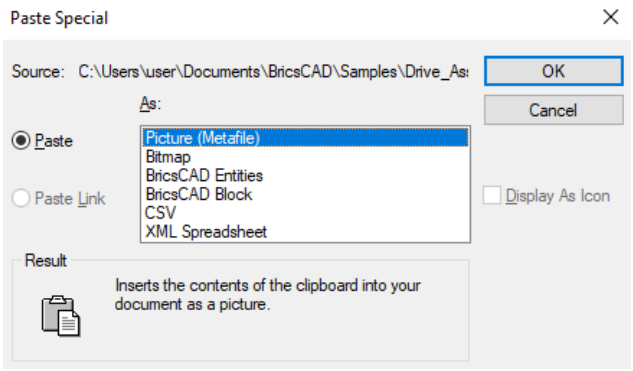
Select mirroring control point *– (Pick a second point for the mirror line)*

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.

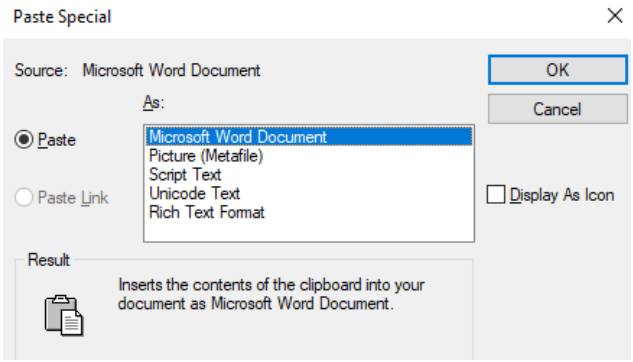
Paste As BricsCAD Block

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:



21.26.4 Paste Link

This option is not available for CAD entities. Pastes the entities in the drawing as an OLE object and then creates a link to the originating application.

21.26.5 Result

Displays a short description of the selection option you want to use.

21.26.6 Display As Icon

This option is not available for CAD entities. Displays the pasted entities as an icon identifying their source application.

21.27 PBLOCKOPERATIONSDISPLAY command

Shows/hides parametric block operation definition lines.

Shape Lite Pro Mechanical BIM

Icon:

21.27.1 Method

The command turns On/Off the visibility of parametric block operation definition lines that are attached to the selected entities.

The approach for parametric block operation geometry is that of a non-database-resident geometry, with the geometry being created on an anonymous layer.

Note:

- When opening a new drawing, the local parametric block operation geometry is not visible by default.
- When a parametric operation is copied, the geometry of the new operation is set visible by default.
- When all entities that are affected by a parametric operation are copied, the parametric operation is also copied and set visible by default.
- When a BEDIT session is started, all local parametric block operations geometry is visible by default.

21.27.2 Options within the command

Show
Shows the parametric block operation geometries of the selected entities.

Hide
Hides the parametric block operation geometries of the selected entities.

21.28 PCNEAREST command

Toggles the **Point Cloud Nearest Point** entity snap.

Shape Lite Pro Mechanical BIM

21.28.1 Description

Toggles the **Point Cloud Nearest Point** entity snap to enable or disable snapping to point cloud nearest point.

You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the 3DOSMODE system variable accordingly.

You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the 3DOSMODE system variable.

21.29 PDF command

Opens the **Attachments** panel.

Shape Lite Pro Mechanical BIM

Icon:

21.29.1 Description

Opens the **Attachments** panel to display it in the current workspace. The **Attachments** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Attachments** panel can be either floating, docked or stacked.

21.30 PDFADJUST command

Adjusts graphical properties of a PDF underlay.

Shape Lite Pro Mechanical BIM

Icon: 

21.30.1 Description

Adjusts the face, contrast and monochrome level of one or more PDF underlays attached to the current drawing.

Note: It can be easier to use the **Properties** panel's Underlay Section to adjust the parameters.

21.30.2 Options within the command

Fade

Sets the fade setting of the PDF underlay(s).

- 0: minimum fade: Underlay PDF is fully displayed
- 100: maximum fade: Underlay PDF is barely visible

Contrast

Sets the contrast setting of the PDF underlay(s).

- 0: minimum contrast: both dark and light elements are displayed medium gray
- 100: maximum contrast: dark elements are displayed dark, light elements are displayed light

Monochrome

Toggles the monochrome setting of the PDF underlay(s).

Yes

Turns colors into shades of gray.

No

Keeps the PDF as-is.

21.31 PDFATTACH command

Attaches a PDF file as an underlay.

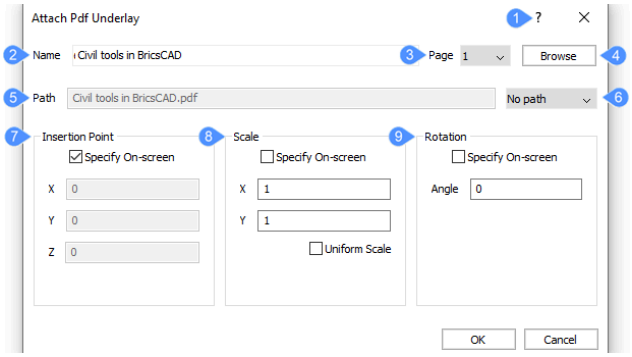
Shape Lite Pro Mechanical BIM

Icon: 

21.31.1 Description

Opens the **PDF Underlay File** dialog box to select a PDF file to reference into the current drawing. After selecting the file, click the **Open** button to display the **Attach PDF Underlay** dialog box. It enables you to specify where and how to attach the PDF file.

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

21.31.2 Help

Opens the Bricsys Help article about the -PDFATTACH command.

21.31.3 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.

21.31.4 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.

21.31.5 Browse

Opens the **Select PDF underlay file** dialog box from where you can select another PDF file than the one already selected.

21.31.6 Path

Displays the path of the file according to the path type you choose.

21.31.7 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.

21.31.8 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.

21.31.9 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.

21.31.10 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.

21.32 -PDFATTACH command

Attaches a PDF file as an underlay via Command line.

Shape Lite Pro Mechanical BIM

21.32.1 Description

Attaches a PDF file as an underlay into the current drawing through the Command line.

21.32.2 Options within the command

PDF Underlay file
Specifies the file name of the PDF file to attach by entering the path of PDF name.

Note: Enter **~** to open the **Select PDF Underlay File** dialog box.

PDF underlay page number
Specifies the page number of a multi-page PDF file to insert.

Note: This option does not appear when the PDF has only one page.

Insertion point
Specifies the insertion point in the drawing. This can be done in 2 ways.

- Picking a point
- Entering x, y coordinates

Scale
Specifies the size of the PDF underlay by entering a scale factor or by moving the cursor.

Size
Sets the size of the insert dynamically. The first point is the insertion point. By moving the cursor, the program ghosts the size and position of the underlay relative to the insertion point.

XY Scale factors
Scales the underlay separately in X and Y direction differently.

Rotation Angle
Specifies the angle of the underlay.

21.33 PDFCLIP command

Clips a PDF underlay to a specified clipping boundary.

Shape Lite Pro Mechanical BIM

Icon: 

21.33.1 Method

Select the PDF underlay and then enter the PDF clipping options.

21.33.2 Options within the command

ON
Turns On the clipping boundary that was turned Off by the **OFF** option.
The area outside the clipping boundary is hidden from the view.

OFF
Turns Off the clipping boundary so the selected PDF is fully visible again. The clipping boundary is saved, until it is deleted, and it can be turned On again with the **ON** option.

Invert
Inverts the clipping mode so that the PDF is clipped either outside or inside the specified boundary.

Note: The **Misc** section of the **Properties** panel also allows you to turn the clipping On and Off, and to invert it.

Delete
Erases the clipping boundary from the selected PDF without prompting or warning you.

New
Allows you to add a clipping boundary to the selected PDF.

Polygonal
Creates non-rectangular clipping boundaries, with a minimum of three sides.

Undo
Undoes the last point.

Rectangular
Creates rectangular clipping boundaries.

21.34 PDFIMPORT command

Imports a PDF and converts its content to simple CAD entities.

Shape Lite Pro Mechanical BIM

Icon:

21.34.1 Method

This command also converts a PDF underlay already attached to the drawing to CAD entities.

Different conversions are made depending on the type of the PDF entity.

PDF entity	CAD Entity
Straight and curved lines	Polylines and splines
SHX text	Polylines

TrueType text	MText
Solid-filled areas and wide lines	Hatches with 50% transparency
Raster images	Raster images
Layers	Layers
Lines with linetypes	Polyline segments

There are 2 methods to convert a PDF to CAD entities

- PDF Underlay
- PDF file

21.34.2 Underlay options

Specify area
Picks the first two points to convert a rectangular area of an underlay to CAD entities.

Polygonal
Picks three or more points to convert a polygonal area of an underlay to CAD entities.

Note: When picking points that are not on the PDF, BricsCAD ignores those points so more points need to be picked.

All
Selects the entire PDF underlay for conversion to CAD entities.

Settings
Displays the PDF Import Settings section of the Setting dialog box.

Keep
Keeps the PDF underlay in place, in addition to converting its vector portions to CAD entities.

Detach
Detaches the PDF underlay so that it is no longer seen in the drawing but is still attached to the drawing.

Unload
Unloads the PDF underlay so that it is no longer seen the drawing but is still attached to the drawing.

21.34.3 Import file Options

Page number
Imports a specific page from the PDF file. When pressing enter, the first page will be imported.

?
Lists the page number in the PDF document.

Note: To view the content of the pages before importing them, use the File Managers preview window.

Settings
Displays the PDF Import Settings section of the Settings dialog box.

Insertion point
Specifies the insertion point in the drawing.

Scale
Specifies the scaling factor or the inserted geometry.

Rotation
Specifies the rotation angle.

Note: The angle needs to be entered and cannot be specified by picking points in the drawing.

21.35 -PDFIMPORT command

Imports a PDF and converts its content to simple CAD entities.

Shape Lite Pro Mechanical BIM

For more information see PDFIMPORT command.

21.36 PDFLAYERS command

Toggles the display of layers in PDF underlays.

Shape Lite Pro Mechanical BIM

21.36.1 Description

Toggles the display of layers in PDF underlays. After entering the command and selecting a PDF underlay, the **Underlay Layers** dialog box displays. Here you can edit the display of the layers of the PDF.

21.37 PDFOPTIONS command

Opens the **Settings** dialog box with the **PDF export** category expanded.

Shape Lite Pro Mechanical BIM

21.37.1 Description

Opens the **Settings** dialog box with the **PDF export** category expanded to view and modify relevant system variables.

21.38 PDIM command

Creates multiple types of dimensions as ACM entities in a single workflow. It is an AMPOWERDIM command.

Shape Lite Pro Mechanical BIM

Icon: 

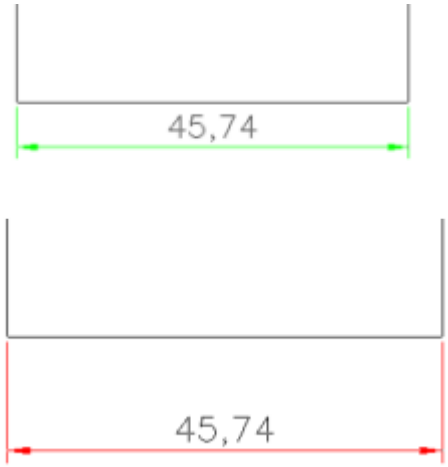
Alias: AMPOWERDIM

Note: This command only works for drawings that contain mechanical entities. To set the environment for BricsCAD Mechanical drawings, see the **Overview of 2D Mechanical Drawings commands** article.

21.38.1 Method

Select two points to create a linear dimension, or select an entity and specify the point for its location. Depending on where you move the dimension line in the drawing, it snaps from horizontal, vertical, or aligned dimensions.

When SNAP is ON, the PDIM dimension line freezes during placement to the snap distance and is marked with red color.



Then, the **Edit Dimensioning** dialog box opens to define the dimension's parameters. For more information about the dialog box, see the related article **Edit Dimensioning dialog box**.

Note: The AMPOWERDIMDISPLAY system variable controls the opening of the **Edit Dimensioning** dialog box after placing a power dimension.

21.38.2 Options within the command

Linear
Creates a linear dimension. Launches DIMLINEAR command.

Note: This is the default option.

Angle
Defines the angle for dimension text. Also, you can launch the AMPOWERDIM_ANG command to get this type of dimension directly.

Text
Overrides the automatic dimension text.

Horizontal
Creates a horizontal dimension. Also, you can launch the AMPOWERDIM_HOR command to get this type of dimension directly.

Vertical
Creates a vertical dimension. Also, you can launch the AMPOWERDIM_VER command to get this type of dimension directly.

Rotated
Creates a rotated dimension. It measures the distance between two points in a direction that is at an angle to a line connecting the two points. Also, you can launch the AMPOWERDIM_ROT command to get this type of dimension directly.

Aligned
Creates an aligned dimension. Launches DIMALIGNED command. Also, you can launch the AMPOWERDIM_ALI command to get this type of dimension directly.

Angular
Creates an angular dimension. Launches DIMANGULAR command. Also, you can launch the AMPOWERDIM_ANG command to get this type of dimension directly.

Radial
Creates a radial dimension.

Diameter
Creates a diameter dimension. Launches DIMDIAMETER command. Also, you can launch the AMPOWERDIM_DIA command to get this type of dimension directly.

Radius
Creates a radial dimension. Launches DIMRADIUS command. Also, you can launch the AMPOWERDIM_RAD command to get this type of dimension directly.

Placement options
Specifies the object to include for distance snap calculation.

Options
Opens the **Placement Options** dialog box to set the snap value.

21.39 PEDIT command

Edits polylines.

Shape Lite Pro Mechanical BIM

Icon:

Alias: EDITPLINE, PE

21.39.1 Description

Edits polylines, 3D polylines, and 3D meshes (short for "polyline edit") and converts 2D entities into polylines.

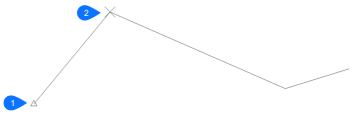
Note: The options of this command change, depending on the entity you are editing. You can edit single or multiple 2D polylines, single 3D polylines, 3D meshes and single or multiple lines, arcs, circles, splines or helixes.

21.39.2 Options within the command

Select polyline to edit
Allows you to select a polyline to edit.

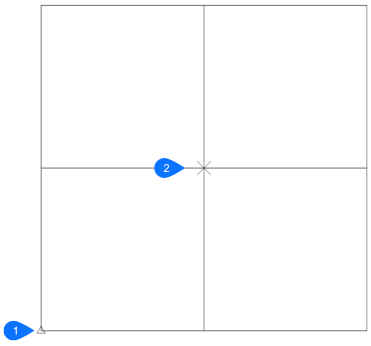
Multiple
Allows you to select multiple polylines to edit.

Edit vertices
Edits the vertices.



1 Triangle marker at the start of the polyline.

2 X marker at the current vertex.



1 Triangle marker at the home vertex.

2 X marker at the current vertex.

Note: It can be easier to edit the vertices with grips editing.

Next vertex
Moves the X marker to the next vertex. When the marker reaches the last vertex of a closed polyline, it does not move to the "next" (first) vertex.

Previous vertex
Moves the X marker to the previous vertex.

Angle
Changes the angle of the current segment. If it is an arc segment, the angle (curvature) of the arc is changed. If it is a line segment, it is turned into an arc. To convert an arc into a line segment, enter an angle of zero degrees.

Note: Angle > 0 is counterclockwise, angle = 0 is straight, angle < 0 is clockwise.

Break
Removes segments from the polyline between the two vertices marked with X markers. This action turns closed polylines into open polylines and open polylines into two polylines, with a gap between them.

Next
Chooses the next vertex.

Previous
Chooses the previous vertex.

Select
Moves second X marker directly to the selected vertex.

Go
Break the polyline between the first and second marked vertices.

Note: It is much easier to use the BREAK or TRIM commands to remove part of a polyline.

Insert vertex
Inserts a vertex. A drag line previews the location of the new vertex.

Left
Moves the X marker to the vertex on the left.

Note: This option is available only for meshes.

RIght
Moves the X marker to the vertex on the right.

Note: This option is available only for meshes.

Up
Moves the X marker to the higher-numbered vertex.

Note: This option is available only for meshes.

Down
Moves the X marker to the lower-numbered vertex.

Note: This option is available only for meshes.

Move
Moves the current vertex. The drag line previews the new location of the vertex.

Note: It is much easier to move the vertices of a polyline with grips editing.

Regen
Regenerates the polyline to show changes made by the editing actions of this command.

SElect
Moves the X marker directly to another vertex that you select. This works faster than using the **Next** and **Previous** options.

Straighten
Removes all segments between two vertices. When you select the start and end points of a multi-segment polyline, the entire polyline is straightened, becoming a single segment; this also applies to closed polylines.

Next
Moves the second X marker to the next vertex.

Previous
Moves the second X marker to the previous vertex.

Select
Moves second X marker directly to another selected vertex.

Go
Replaces the segments between the two marked vertices with a single segment.

Tangent
Edits the tangent direction of the vertex.

Width
Changes the width of the current segment. The current segment is the segment that lays between the X-marked vertex and the next vertex.

Close
Closes the polyline by adding a segment between the start and endpoints. When the polyline is closed, this option reads **Open**.

Open
Opens closed polylines by removing the last segment drawn to close the polyline. When the polyline is open, this option reads **Close**.

Decurve
Reverses the effects of the **Fit** and **Spline** options.

Fit
Fits a curve to the polyline.

Note: To unfit the polyline, use the Decurve option.

Join
Adds open entities to the selected polyline.

Note: This option works only with open and with cojoined entities. Cojoined means that the endpoints of entities meet, such as when drawn with ENDpoint object snap or with the Last point option. The joined entities take on the source polyline's properties, such as color, width, and layer.

Fuzz distance
When endpoints are within the fuzz distance, the entities are extended or trimmed.

Jointype
Specify the type of joining.

Extend
Extends or trims segments to the nearest endpoints.

Add
Adds linear segments between the nearest endpoints.

Both
Extends or trims if possible, otherwise adds linear segments.

Linetype mode
Determines how linetypes appear on polylines.

On
Linetypes start and stop at the polyline's start and endpoints.

Off
Linetypes start and stop at each vertex.

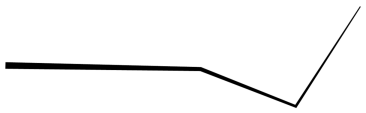
Note: This option is stored in the PLINEGEN variable.

Reverse direction
Reverses the direction of the polyline: the starting point is now its end point, and vice versa. The polyline does not change its look with this operation, except that the triangle marker moves to the other end of open polylines. This option affects operations that depend on the direction of a polyline, such as vertex editing.

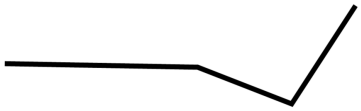
Spline
Converts the polyline into a spline.

Note: All width information is lost. Use the **Width** option to reapply width. The spline is a Bezier-spline whose smoothing is defined by the SPLINETYPE system variable.

Taper
Tapers the width of the entire polyline from one end to the other.



Width
Changes the width of all segments.



Note: This option overrides tapered widths.

Desmooth
Removes the Bezier-spline smoothing of 3D meshes.

M close
Closes the 3D mesh in the m direction. When the mesh is closed, this option reads **M open**.

M open
Opens the 3D mesh in the m direction. When the mesh is open, this option reads **M close**.

N close
Closes the 3D mesh in the n direction. When the mesh is closed, this option reads **N open**.

N open
Open the 3D mesh in the n direction. When the mesh is open, this option reads **N close**.

Undo
Undoes the last action.

eXit
Exits the command.

21.40 PEDITEXT command

Edits vertices and segments of a polyline.

Shape Lite Pro Mechanical BIM

Icons: 

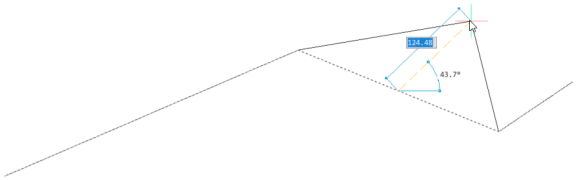
21.40.1 Description

Edits vertices and segments of a polyline interactively, without first selecting the polyline.

Note: This command was meant for use with the Quad cursor. See more information in the **Editing polylines** article, under the **Using the edit polyline segments tools in the Quad** section.

21.40.2 Options within the command

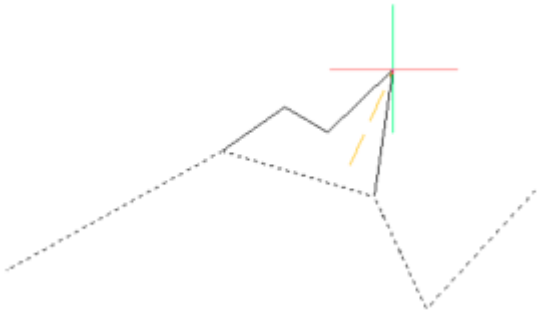
Add vertex
Adds a vertex to the selected segment.



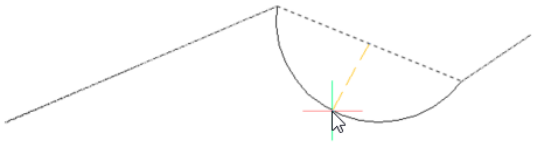
add vertex at the End
Adds a new vertex at the end of the polyline.



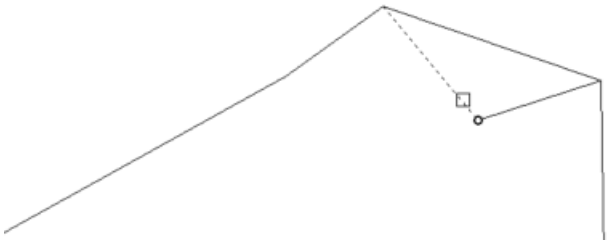
Add Multiple vertices
Adds multiple vertices to the polyline in one session.



adjust Bulge
Modifies the bulge factor of the selected segment.



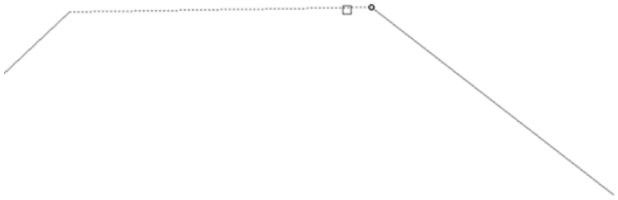
Remove segment
Deletes the selected segment. A preview result is displayed.



Remove Multiple segments
Deletes multiple segments of the polyline in one session. A preview result is displayed.

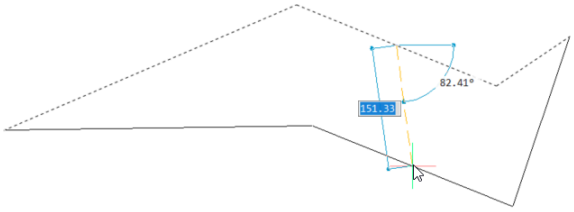
remove vertex (D)
Removes a vertex. Activate the vertex by moving the cursor near the desired end of the segment.

Note: A dot will mark the active vertex.

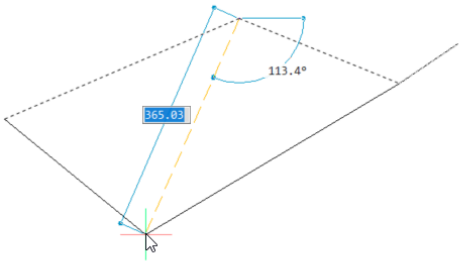


remove multiple vertices (DM)
Removes multiple vertices of the polyline in one session. A preview result is displayed.

Stretch segment
Moves the selected segment.



Stretch Vertex
Moves the selected vertex.



21.41 PERPENDICULAR command

Toggles the **Perpendicular** entity snap.

Shape Lite Pro Mechanical BIM

Icon:

21.41.1 Description

Toggles the **Perpendicular** entity snap to enable or disable snapping to perpendicular. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the

OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

21.42 PFACE command

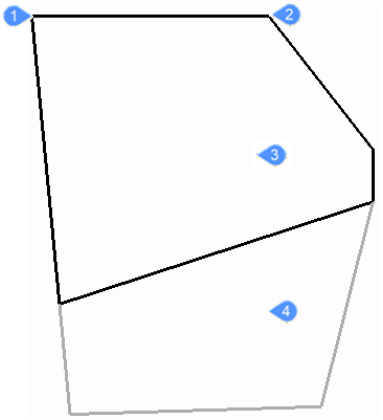
Draws multi-sided meshes.

Shape Lite Pro Mechanical BIM

Icon:

21.42.1 Description

Draws multi-sided polyface meshes in 3D; meant for use by macros.



1 Vertex 1

2 Vertex 2

3 Face 1

4 Face 2

21.42.2 Options within the command

Invisible
(Hidden option.) Specify an invisible edge by entering a negative number.

Color
Specify the color of the edge by entering a color name, number, or color book name.

Layer
Specify the layer name for the edge by entering the name of a layer.

21.43 PLACEVIEW command

Imports named views from a source drawing.

Shape Lite Pro Mechanical BIM

21.43.1 Description

Places named views from a source drawing into a paper space layout of the current drawing.

Note:

- This command works only in paper space, and is meant for use with sheet sets.
- It is not allowed to place a named view in a layout of the source drawing.

21.43.2 Method

There are two methods to place named views:

- Command line: start the command by typing placeview in the Command line.
- Mouse: drag a model view from the **Content Browser** panel onto a paper space layout.

21.43.3 Options within the command

Enter file name
Allows you to enter the file name from the source file from which the views will be imported.

Enter view name to place
Allows you to enter the name of the view you want to import.

?
Prints a list of the named views in the Prompt History window.

Enter view origin point
Allows you to specify the location of the lower left corner of the named view.

21.44 PLAN command

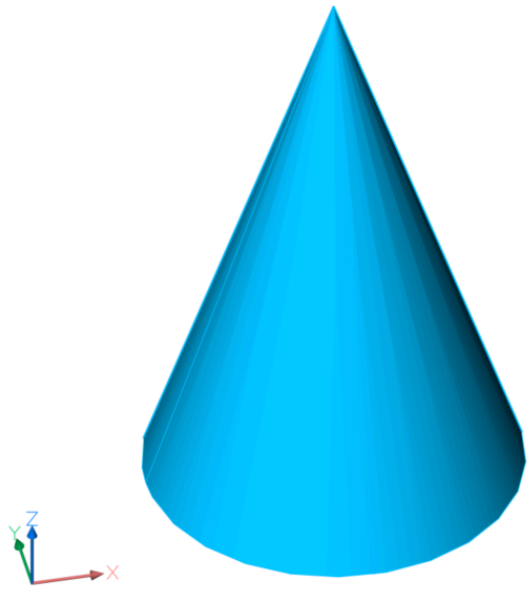
Displays the plan viewpoint of drawings.

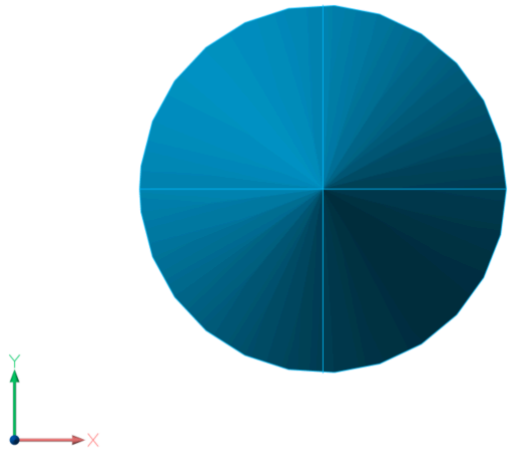
Shape Lite Pro Mechanical BIM

Icon: 

21.44.1 Description

Shows the 3D model in plan view which looks straight down on the x,y-plane.





21.44.2 Options within the command

UCS
Displays the plan view of a named UCS or Dynamic UCS after entering a name or placing the cursor over a 3D solid.

Note: To be able to run this command the second way, make sure that the UCSDETECT system variable is turned On.

?
Lists the names of UCSs in the current drawing.

World
Displays the plan view of the World Coordinate System.

Current
Displays the plan view of the current UCS.

21.45 PLINE command

Creates a polyline.

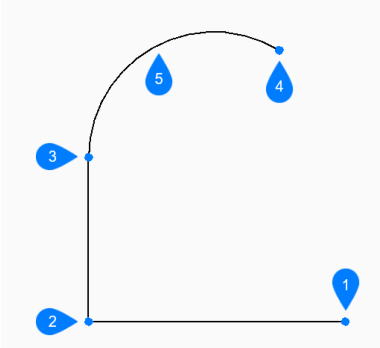
Shape Lite Pro Mechanical BIM

Icon:

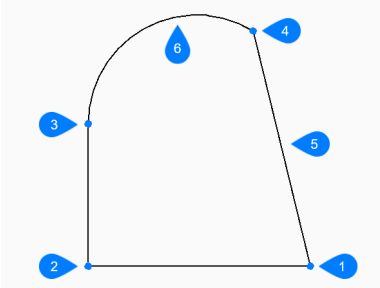
Alias: PL, POLYLINE

21.45.1 Description

Creates a single polyline entity with multiple line and arc segments by specifying the start and end point of each segment. Options allow you to switch between line and arc segments, apply widths, undo, and close the geometry.



1 Start
2 Next
3 Next
4 End
5 Arc segment



1 Start/End
2 Next
3 Next

4 Next

5 Close

6 Arc segment

21.45.2 Method

This command has 3 methods to begin creating a polyline:

- Start of polyline
- Last point
- Follow

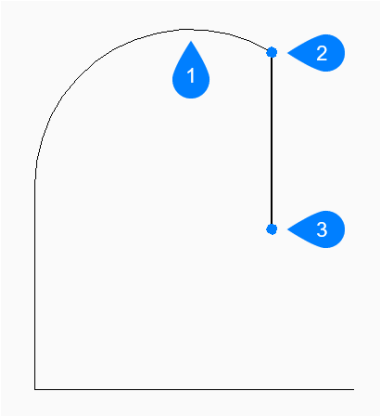
21.45.3 Options within the command

Start of polyline
Allows you to begin creating a polyline by specifying a start point.

Set next point
Specifies the next vertex of the polyline.

Note: You can continue adding unlimited segments until you press **Enter** to end the command.

Last point
Begins creating a polyline from the last point picked.



1 Last drawn segment

2 Last point

3 End point

draw Arcs
Draws polyline arc segments by specifying the end of arc.
The arc is drawn tangent to the previous segment.

Angle
Specifies the included angle of the arc segment.

CEnter
Specifies the center point of the arc segment.

CLose
Automatically draws a polyline arc segment from the end point of the last segment to the start point of the first segment. This ends the command.

Direction
Specifies the direction of the arc segment.

Radius
Specifies the radius of the arc segment.

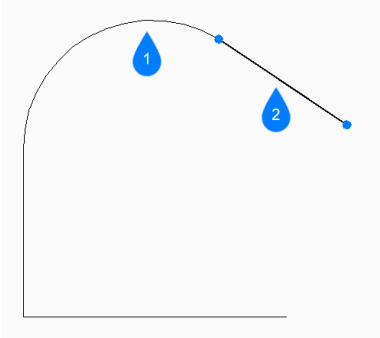
Second point
Specifies a point along the arc's circumference.

draw Lines
Draws line segments.

Distance
Specifies the distance for the line segment.

Angle of segment
Specifies the angle of the line segment.

Follow
Begins creating a polyline from the last drawn arc or line segment, following its angle.



1 Last drawn segment

2 Follow

Halfwidth
Specifies the starting and ending half-width of the segment, as well as all segments that follow -- until you change the width or halfwidth.



1 Halfwidth

Width
Specifies the starting and ending width of the segment, as well as all segments that follow -- until you change the width or halfwidth.



1 Start width

2 End width

Undo
Undoes the last polyline segment and continue drawing from its previous start point.

Close
Automatically draws a polyline segment from the end point of the last segment to the start point of the first segment. This ends the command.

21.46 PLOT command

Plots drawings to printers and to files.

Shape Lite Pro Mechanical BIM

21.46.1 Description

Opens the **Print** dialog box to specify print options and preview or print the current drawing.

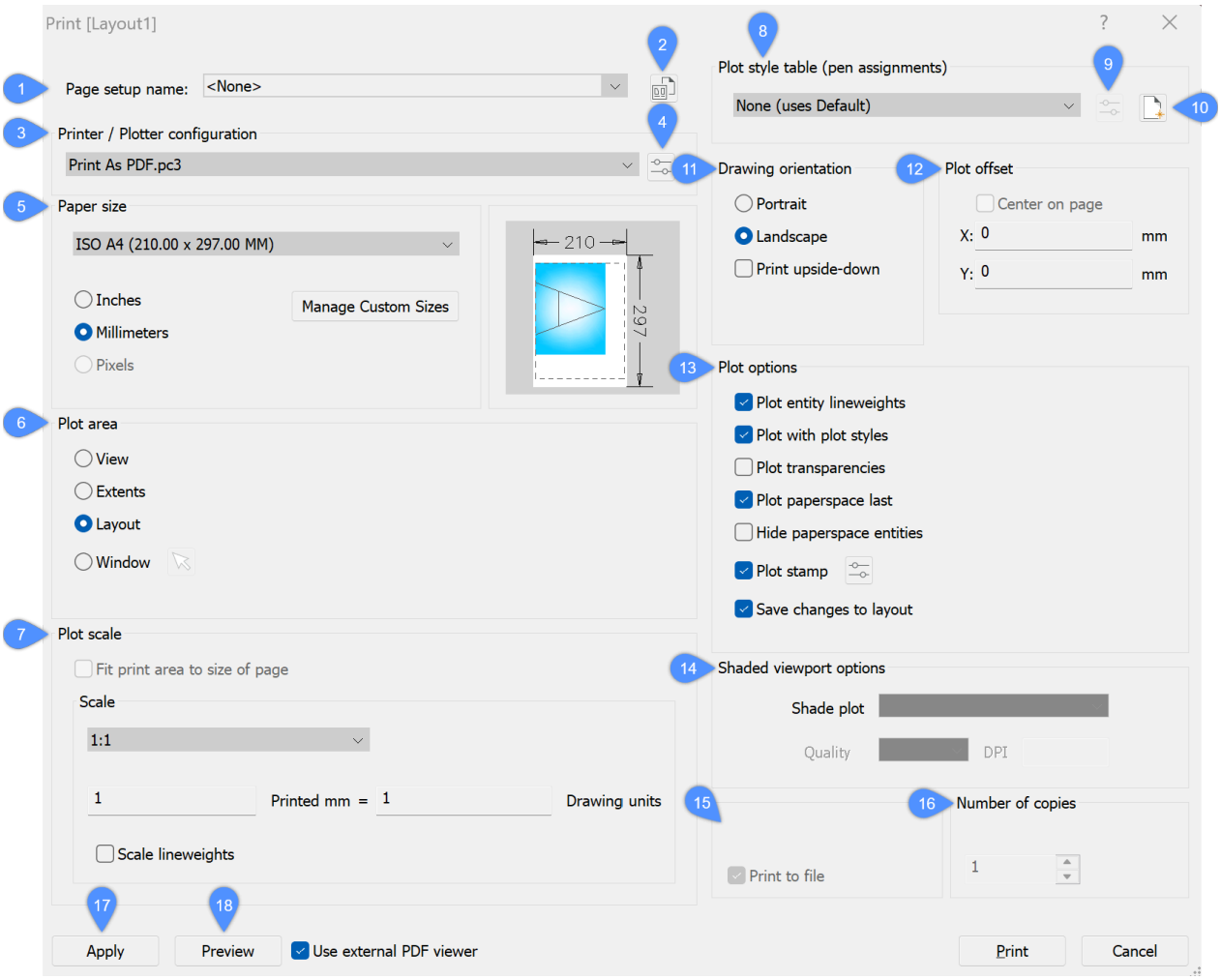
Note: This command adds a watermark to print output obtained using an Academic license.

Remember: On macOS and Linux it is not possible to print to system printers. It is only possible to print to the **Print As PDF.pc3** printer. So, a PDF document will be generated, which needs to be sent to the printer for physical printing.

The **Print** dialog box allows you to print and preview drawings to plotters and files.

Remember:

- On macOS and Linux it is not possible to print to system printers. It is only possible to print to the **Print As PDF.pc3** printer. So, a PDF document will be generated, which needs to be sent to the printer for physical printing.
- On macOS and Linux there is a relevant PAGESETUP command.



1 Page setup name
2 Create page setup
3 Printer/Plotter configuration
4 Edit plotter configuration
5 Paper size
6 Plot area
7 Plot scale
8 Plot style table
9 Edit Plot Style
10 Create New Plot Style
11 Drawing orientation
12 Plot offset
13 Plot options
14 Shaded viewport options

15 Print to file

16 Number of copies

17 Apply

18 Preview

21.46.2 Page setup name

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.

21.46.3 Create page setup

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

21.46.4 Printer/Plotter configuration

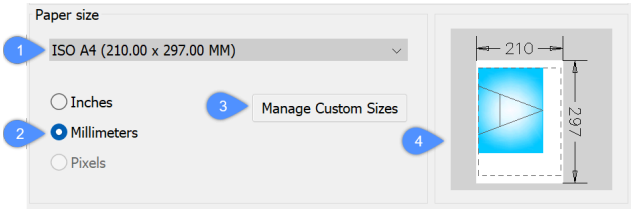
Allows you to select from the drop-down list the printer or plotter.

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.

21.46.5 Edit plotter configuration

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3 files. See the PLOTTERMANAGER command.

21.46.6 Paper size



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.

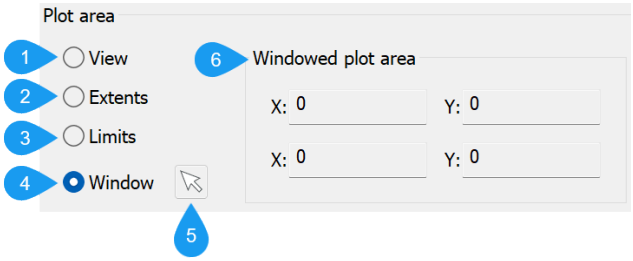
Manage Custom Sizes

Note: This option is available for the PDF, PNG, TIF, BMP, and JPG outputs.

Manages custom paper sizes for the PC3 printer selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.
You can attach a predefined plotter model parameter (PMP) file to a plotter configuration (PC3) file.

Preview
This section indicates the size, position and orientation of the current plot area on the selected paper size.

21.46.7 Plot area



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.

21.46.8 Plot scale

Scales the drawing to fit it to the paper:

- **Fit print area to size of page**: 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.

21.46.9 Plot style table

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.

21.46.10 Edit Plot Style

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

21.46.11 Create New Plot Style

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).

21.46.12 Drawing orientation

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.

21.46.13 Plot offset

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.

21.46.14 Plot options

- **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.

21.46.15 Shaded viewport options

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.

21.46.16 Print to file

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

21.46.17 Number of copies

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

21.46.18 Apply

Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

21.46.19 Preview

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.

21.47 -PLOT command

Plots drawings to printers and to files.

Shape Lite Pro Mechanical BIM

21.47.1 Description

Plots drawings to printers and to files, via the Command line.

Note: This command is meant for scripts and routines.

Note: This command adds a watermark to print output obtained using an Academic license.

21.47.2 Method

Choose whether to proceed a detailed plot configuration.

21.47.3 Options within the command

Detailed plot configuration?
Choose whether to change the plot configuration.

Yes
Prompts 16 options to configure the plot.

No
Prompts 6 options.

Enter a layout name or ?
Specifies the name of the layout to plot. Enter ? to list the layouts in the current drawing.

Enter an output device name or ?
Specifies the name of the output device (plotter or file). Enter ? to list the available plot devices.

Enter paper size or ?
Specifies the name of the paper size. Enter ? to list the available media sizes.

Enter paper units
Specifies the paper units: inches or millimeters.

Enter drawing orientation
Specifies the drawing orientation: portrait or landscape.

Plot upside down?
Plots the drawing upside down.

Note: This option is useful when paper with a drawing border is loaded backwards in the printer.

Enter plot area
Specifies which area of the drawing to plot.

Display
Plots the area shown in the current viewport.

Extents
Plots the extents of the drawing, ensuring all non-frozen entities are plotted.

Limits
Plots the limits of the drawing, as defined by the LIMITS command.

View
Plots the current view, or a named view. See the VIEW command.

Window
Plots a rectangular area defined by the user.

Enter plot scale
Specifies the plot scale factor.

Note: The **Fit** option calculates the scale factor automatically.

Enter plot offset
Specifies the offset distance for the print.

Note: 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 and negative values move the drawing down and to the left.

Note: The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

Plot with plot styles?
Toggles the use of plot styles.

Note: Plot styles override lineweight settings.

Enter plot style table name
Specifies the plot style table to use, which assigns properties to "pens", colors, and entities.

Note: Enter ? to list the plot styles available in the drawing.

Plot with lineweights?
Toggles the use of lineweights.

Note: This option is unavailable when **Plot with Plot Styles** is turned on.

Yes
Mimics lineweights assigned to the drawing in the print.

Enter shade plot
Specifies the shade plot. You can choose between: **As displayed**, **legacy Wireframe**, **legacy Hidden**, **Rendered**, **Visual style**, and **Unchanged**. Overrides the visual style of the current view when printing model space.

Note: 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.

Write plot to a file?
Determines whether the plot is sent to a file. If Yes, displays the **Create plot file** dialog box.

Note: The plot is saved in a .plt file.

Save changes to layout?
Toggles saving the plot parameters to the model or layout tab.

Proceed with plot?
Toggles plotting of the drawing.

Enter a page setup name
Specifies the name of a page setup.

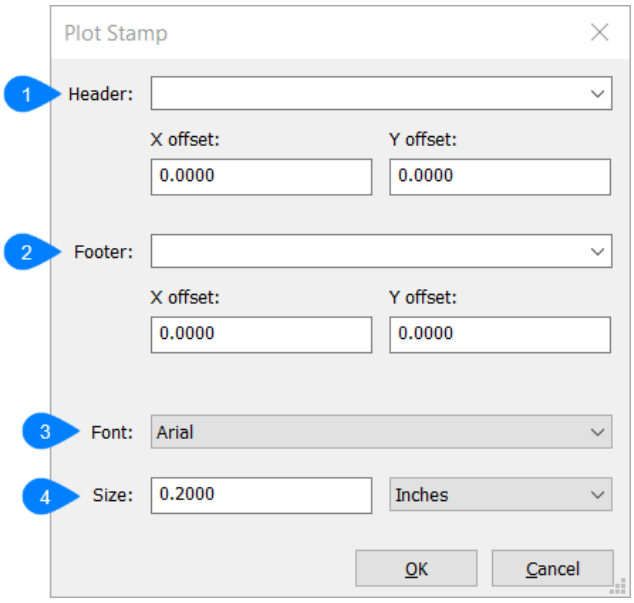
21.48 PLOTSTAMP command

Opens the **Plot stamp** dialog box.

Shape Lite Pro Mechanical BIM

21.48.1 Description

Opens the **Plot Stamp** dialog box to add information about drawings to the edge of the plotted drawing.



1 Header

2 Footer

3 Font

4 Size

21.48.2 Header

Appears at the top of the plot. A meta-phrase for the header can be chosen by clicking the drop-down list. You can also specify the X and Y offsets for the header.

21.48.3 Footer

Appears at the bottom of the plot. A meta-phrase for the footer can be chosen by clicking the drop-down list. You can also specify the X and Y offsets for the footer.

21.48.4 Font

Specifies the font used for the plotstamp.

21.48.5 Size

Specifies the height of the text.

- Inches: specifies the height of the text in inches.
- Millimeters: specifies the height of the text in millimeters.

21.49 PLOTSTYLE command

Sets the current plot style.

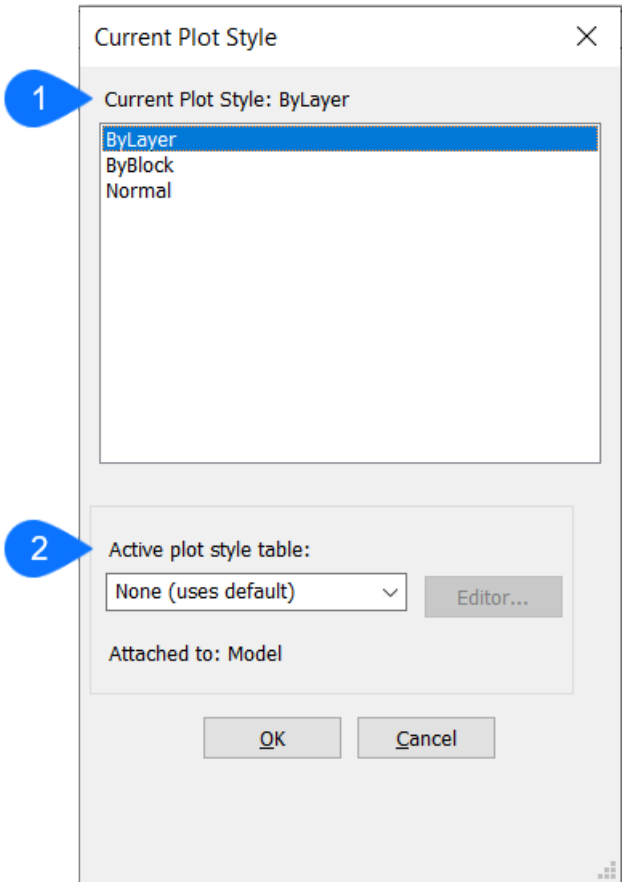
Shape Lite Pro Mechanical BIM

21.49.1 Description

The command opens the **Current Plot Style** dialog box.

Note: This command operates only when plot styles are enabled in drawings.

The **Current Plot Style** dialog box allows you to set the current plot style.



1 Current Plot Style

2 Active plot style table

21.49.2 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.

21.49.3 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.

21.50 PLOTTERMANAGER command

Create and edit plotter settings.

Shape Lite Pro Mechanical BIM

21.50.1 Description

Opens a generic **PlotConfig** system dialog box where you can choose a plotter configuration file or where you can create and edit customized parameter PC3 files for printers and other output devices.

21.51 PLT2DWG command (Express Tools)

Imports HPGL plot files into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

21.51.1 Method

Opens the **Select plot file to import** dialog box that allows you to choose a PLT file to open.

21.52 PNGOUT command

Saves the entities from the current drawing to a PNG file.

Shape Lite Pro Mechanical BIM

21.52.1 Description

Opens the **Save Png** dialog box to save the entities from the current drawing to a PNG file. The current view specifies what entities to include in the image.

21.53 POINT command

Creates a point.

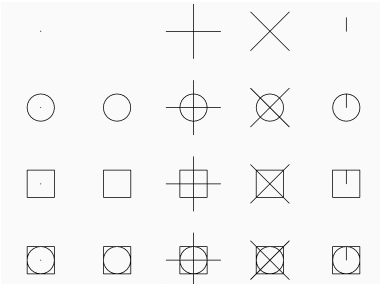
Shape Lite Pro Mechanical BIM

Icon:

Alias: PO

21.53.1 Description

Creates one or multiple points, controlling their size and style.



21.53.2 Method

Create a single point, represented by a dot, by specifying the location of point.

21.53.3 Options within the command

Location of point
Specifies the location of the point.

point Settings...
Opens the **Settings** dialog box to specify the display mode and size for points.

Multiple points
Draws multiple points until you press **Enter** key to end the command.

21.54 POINTCLOUD command

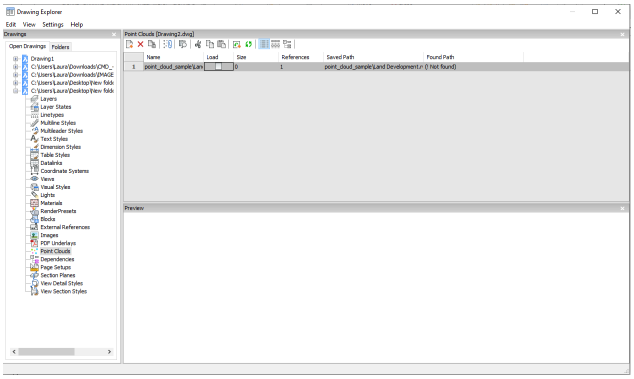
Manages Point Cloud files.

Shape Lite Pro Mechanical BIM

Icon:

21.54.1 Description

Displays the **Point Clouds** category of the **Drawing Explorer**:



21.54.2 Option within the Drawing Explorer

Name
Reports the name of the Point Cloud file insert.

Load
Toggles loading of the Point Cloud file.

- On: loads the Point Cloud, and makes it visible
- Off: unloads the Point Cloud, making it invisible

Size
Reports the size of the Point Cloud file. Very large files can slow down the system.

References
Reports the number of times the Point Cloud is attached to the drawing.

Saved Path
Reports the original path to the Point Cloud file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing Point Cloud file.

Found Path
Reports the current path to the Point Cloud file. This path should match the Saved Path in most cases.

21.54.3 Context menu options

New
Attaches a Point Cloud file to the current drawing, like the POINTCLOUDATTACH command.

Delete
Removes the selected Point Cloud from the current drawing.

Insert
Inserts additional copies of the existing Point Cloud files into the drawing; displays the **Attach Point Cloud** dialog box, like the POINTCLOUDATTACH command.

Select all
Selects all Point Clouds.

Invert selection
Deselects the current selection and vice versa.

Run extended search for missing attachments
Runs an extended search for missing attachments.

21.55 POINTCLOUDALIGN command

Rotates the point cloud to optimally align it with the X and Y axes.

Shape Lite Pro Mechanical BIM

Icon: 

21.55.1 Description

Rotates the point cloud to optimally align it with the X and Y axes in the world coordinate system or creates an UCS aligned with the point cloud. This enables you to efficiently create and edit entities relative to the point cloud using common tools such as orthogonal and polar tracking.

21.55.2 Options within the command

Select point cloud entity
Allows you to select a point cloud entity if the current drawing has more than one point cloud attached.

Note: If the current drawing has only one point cloud attached, it is automatically selected for the align operation.

First corner point
Specifies the first corner of a rectangular selection window.

Specify opposite corner
Specifies the opposite corner of a rectangular selection window.

Ucs
Creates an UCS aligned with the point cloud.

Yes
Saves the UCS using the point cloud name or a user-specified name.

No
UCS is created without saving.

Rotate
Rotates the point cloud to optimally align it with the X and Y axes in the world coordinate system.

21.56 POINTCLOUDATTACH command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

21.56.1 Method

Select a file or a folder, specify the point cloud data name and, if needed, enter a unit for point coordinates. Then, attach it from the **Attach Point Cloud** dialog box. See also the **Attach Point Cloud dialog box** article.

Note:

- The supported files types are: *.e57, *.hspc, *.las, *.laz, .lgsx, *.pack, *.pts, *.ptx, *.rcp, *.rcs, *.rdbx, *.rsp, *.zfs.
- For MacOS/Linux, the import of *.rsp and *.rdbx files is not yet supported.

The point cloud file or folder is preprocessed in the background to create a cache entry, and a balloon message displays when the point cloud is fully cached.

Note:

- The double-byte characters are accepted for the point cloud name.
- When the same source data is selected again for which cache is already available, this is detected and the point cloud is directly inserted in the DWG file without new preprocessing.
- The POINTCLOUDCACHEFOLDER system variable defines the folder where the point cloud data are stored.

- When the POINTCLOUDHSPC system variable is set to ON, the file is inserted in the HSPC format.
- When a LAS/LAZ file contains classification data, it is shown in the **Point Cloud Manager** panel. Visibility of the classified points can be toggled on or off per class. This information is only available if the point cloud is preprocessed in HSPC format.
- When geographic information is available in the original data, it is stored in the point cloud's cache (only if the point cloud is preprocessed in HSPC format). You can access this information in the **Properties** panel under the **Misc** category for a selected point cloud instance. To overwrite the DWG file's geographical information with that of the point cloud instance (possible only if the CRS of the point cloud is the same as the CRS of the DWG file), turn On the **Geolocate** property of the selected point cloud instance.

21.56.2 Options within the command

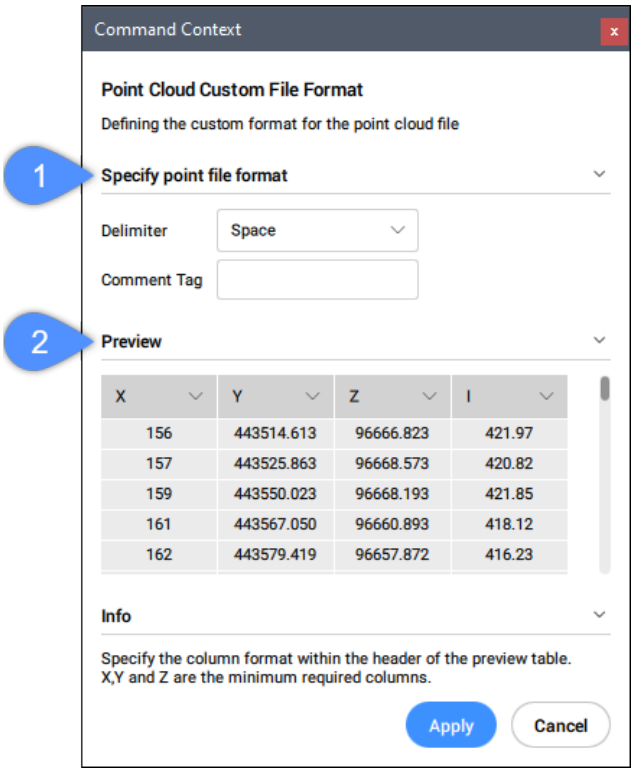
File

Opens the **Pre-process Point Cloud Data File(s)** dialog box from where you have to select a point cloud file to attach. See also the **Pre-process Point Cloud Data File(s) dialog box** article.

Note:

- The source file name is the same as the cache key and is used to check if the point cloud is available in the cache folder.
- The point cloud cache name is a user specified name for the preprocessed point cloud. It is the default point cloud name if the user does not specify a point cloud name for insertion. Multiple insertions can have the same or a different point cloud name.

Tip: You can import an ASCII point cloud file of almost any extension, with a custom separator, or a custom data distribution across the columns. When an unsupported file format is selected, you can define how the point data is stored in the file by accessing the Command line options or the **Point Cloud Custom File Format** command context panel. To be able to select an unsupported file format, select **all files (*)** in the open file/folder dialog.



Specify point file format (1)
Defines the custom format for the point cloud file.

Delimiter
Sets a delimiter from the drop-down list.

Comment Tag
Defines a comment tag. Any content after this tag in the same line is ignored.

Preview (2)
Displays a preview of the point file. You can configure the column format.

columnFormat
Sets a column format.

fOlder
Opens the **Choose Input Folder** dialog box to select a folder to import.

21.57 -POINTCLOUDATTACH command

Attaches point cloud files to the current drawing via Command line.

Shape Lite Pro Mechanical BIM

21.57.1 Description

Attaches point cloud files that were previously converted to BricsCAD's BPT format to the current drawing.

21.57.2 Options within the command

Inputdata
Allows you to specify the point cloud link to attach to the current drawing.

Enter insertion point
Allows you to enter the insertion point coordinates or specify it on the screen.

Rotation angle
Allows you to enter the rotation angle or specify it on the screen.

Scale factor
Allows you to enter the scale factor or specify it on the screen.

21.58 POINTCLOUDBUBBLEVIEWER command

Opens the **Bubble viewer**.

Shape Lite Pro Mechanical BIM

21.58.1 Description

Opens the **Bubble viewer** for the point cloud bubble with the associated specified index. The name of the selected bubble (active scan) is displayed in the title bar of the viewer.

Note: Bubbles can also be opened in the **Bubble viewer** from the **Point Cloud Manager** panel's **Scans/Waypoints** list via the **Open scan in bubble viewer** option in the context menu of the selected scan/waypoint.

21.58.2 Bubble Viewer

Depending on the original file format of the point cloud and the type of scanner used during scanning (static or kinematic), bubbles (green spheres) or waypoints (blue spheres) might be shown at all the scan locations. At these locations, you will experience the most realistic visual representations by opening the **Bubble viewer**.

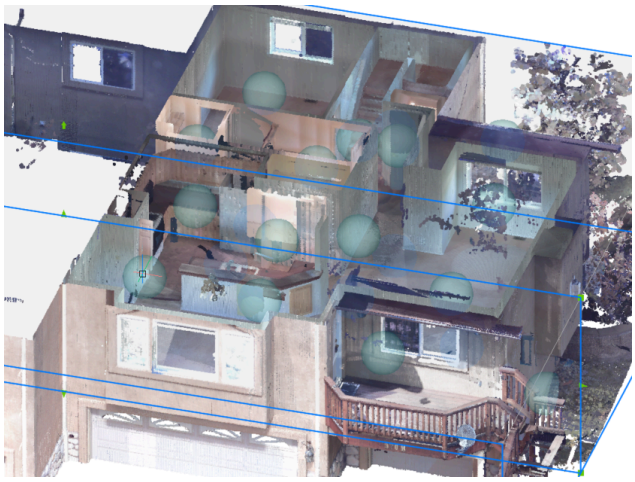
Note: LAS and PTS formats do not contain scanner information, so they cannot produce point cloud bubbles.

Structured data is captured using a static scanner. In this case the location of the scanner is known for every point. Green bubbles are created at the locations of the static scanner from the scanned points.

Note: The bubbles have depth information and can be used for algorithms like fit planar and fit cylinder.

Unstructured data is captured using a kinematic scanner. In this case there is no exact location from where the points were scanned. Some of the kinematic scanners create panoramic images at time intervals. For these images the location is known and waypoints (blue spheres) are created.

Note: The waypoints are purely created from the captured images which don't have depth information. They have a pure visualization functionality and cannot be used for other algorithms.



Note: Change the size of the displayed bubbles from the **Point Cloud Manager**'s **Render Settings** tab. From here you can also turn off the display of point cloud bubbles in Model space and **Bubble viewer**. For cropped point clouds you can choose to display only the non-cropped bubbles (see **Point cloud manager panel** article).

Indicate a bubble index in the POINTCLOUDBUBBLEVIEWER command or double-click one of the bubbles in the model space to open the **Bubble viewer**.

Press and hold the middle mouse button and move the mouse to view the point cloud in any direction from that scan location.

Zoom in and out using the mouse wheel.



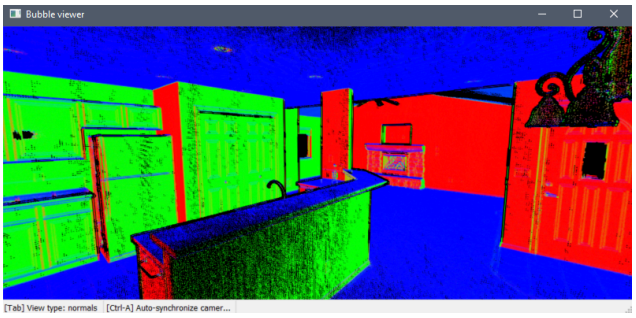


Nearby scans are indicated in the **Bubble viewer** by red bubbles. Hover the cursor over a bubble to display the scan name. Double-click a bubble to open the scan in the viewer.

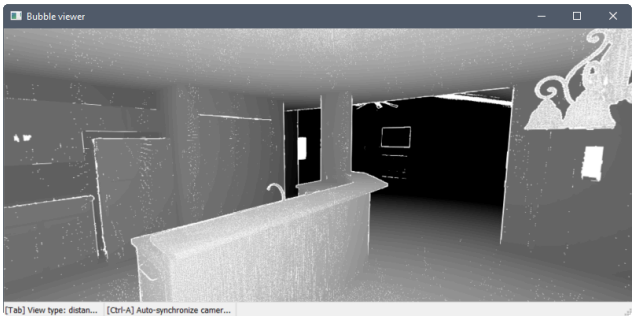


Press the **Tab** key to cycle through three different visual modes (view types).

- The **colors** and **colors (depth)** modes display the points as their actual colors or in grayscale, depending on how the data was scanned.
- The **normals** mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.



- The **distances** mode displays the points from light to dark as the distance from the scan location increases.



Sync the drawing view to match the **Bubble viewer** by pressing **Ctrl+A**.



Note: During commands that prompt for selection of a point, e.g. (poly)lines, you can use the **Bubble viewer** to snap to point cloud points.

21.59 POINTCLOUDCLASSIFY command

Assigns classes to each point in a point cloud.

Shape Lite Pro Mechanical BIM

Icon: 

21.59.1 Requirements

- The Point Cloud Classifier runs on Windows only.
- Your system's GPU needs to be CUDA compatible.
- ***Note:*** You can check if your GPU is CUDA compatible here.
- Make sure that the latest NVIDIA driver is installed and running.

21.59.2 Preparing the classifier libraries

1 Download the classifier libraries from here.

2 Unzip the archive.

3 Copy the content of the folder to the BricsCAD installation folder:

- Select all the DLL files and the **UserDataCache** folder inside the classifier libraries, then drag and drop these over the installation folder. All the DLL files will be copied, and the **UserDataCache** folder will be updated, with the **PCC models** folder copied inside it.
- Alternatively, you can first copy and paste all the DLL files inside the classifier libraries to the installation folder, then copy and paste the **PCC models** folder to the **UserDataCache** folder inside the installation folder.

The BricsCAD installation folder's default path is: *C:\Program Files\Bricsys\BricsCAD V25 en_US*.

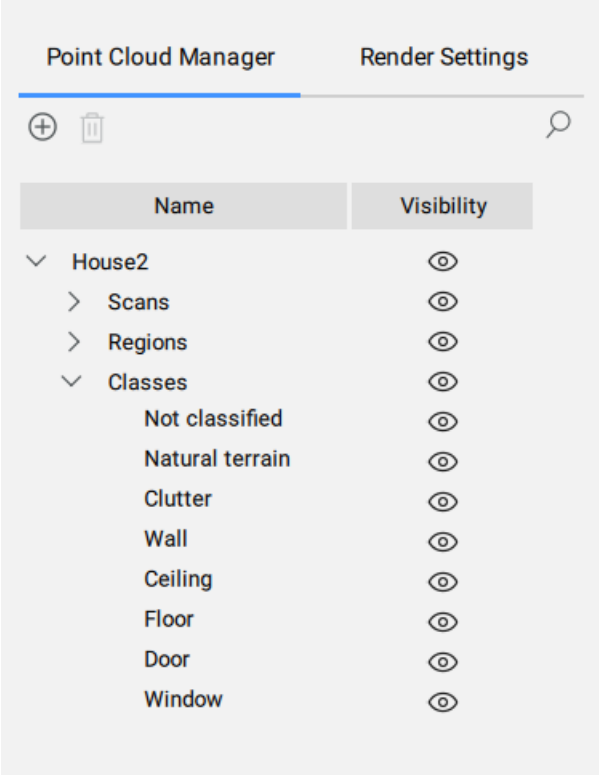
Important: You need to re-run BricsCAD after you copy the classifier libraries.

If all the above requirements are met and libraries correctly copied, the POINTCLOUDCLASSIFY command will run the classifier.

21.59.3 Method

When the command is called and the model is selected, the classifier will run in the background and the results will appear in the **Point Cloud Manager** panel when ready.

Note: Use the POINTCLOUDMANAGERPANELOPEN command to open the **Point Cloud Manager** panel.



There are 4 different models to use with the classifier:

- **indoorBasic**: 10 basic classes for indoor scans (walls, ceiling, floor, windows, doors, clutter, etc.).
- **indoorSimplified**: model with 20 classes for indoor scans (basic classes + Inclined roof, Roof Equipment, Steel structure, Railing, etc.).
- **indoorFull**: model with 40 classes for indoor scans (simplified classes + Pipe installation, Wire installation, Radiator, Plumbing, etc.).
- **Outdoor**: model with 5 classes for outdoor scans (Terrain, Vegetation, Building, Paved road - hardscape, Noise).
- **Roads** (starting with V25.2.03): model with 30 classes, used for classification of point clouds typically captured via a scanner installed on a car (Vegetation, Buildings, Road surface, Curbstone, Guard rails, Sign posts, Vehicles, etc.)
- **Construction** (starting with V25.2.03): model with 15 classes, used for classification of point clouds captured from a construction site (Vegetation, Building, Road and hardscape, Fencing, Construction material, Trenching, etc.)

Note: For indoor models, the results will be more accurate if room information is available. To obtain room information, follow these steps:

1 Run the POINTCLOUDDETECTFLOORS command to detect the floors in the point cloud. The detected floors are listed in the **Point Cloud Manager** panel under **Building**.

2 (Optional) To edit the selection of points assigned to a floor, right-click the floor listed in the **Point Cloud Manager** panel and select the **Modify** option from the context-menu (see also the

POINTCLOUDMODIFYFLOOR command article).

3 Run the POINTCLOUDDETECTROOMS command to detect rooms in the selected floor/s. The results will appear under **Building** in the **Point Cloud Manager** panel.

21.59.4 Options within the command

Select model
Allows you to select one of the classification models.

indoorBasic
Classifies point clouds using basic indoor classes.

indoorFull
Classifies point clouds using full indoor classes.

indoorSimplified
Classifies point clouds using simplified indoor classes.

Outdoor
Classifies point clouds using outdoor classes.

Roads
Classifies point clouds using road classes (starting with V25.2.03).

Construction
Classifies point clouds using construction classes (starting with V25.2.03).

Select classification mode
Allows you to select the preset configuration for the classifier.

Speed
Prioritizes speed over quality in the classification process.

Quality
Prioritizes quality over speed in the classification process.

21.60 POINTCLOUDCOLORMAP command

Colorizes the point cloud.

Shape Lite Pro Mechanical BIM

Icon: 

21.60.1 Description

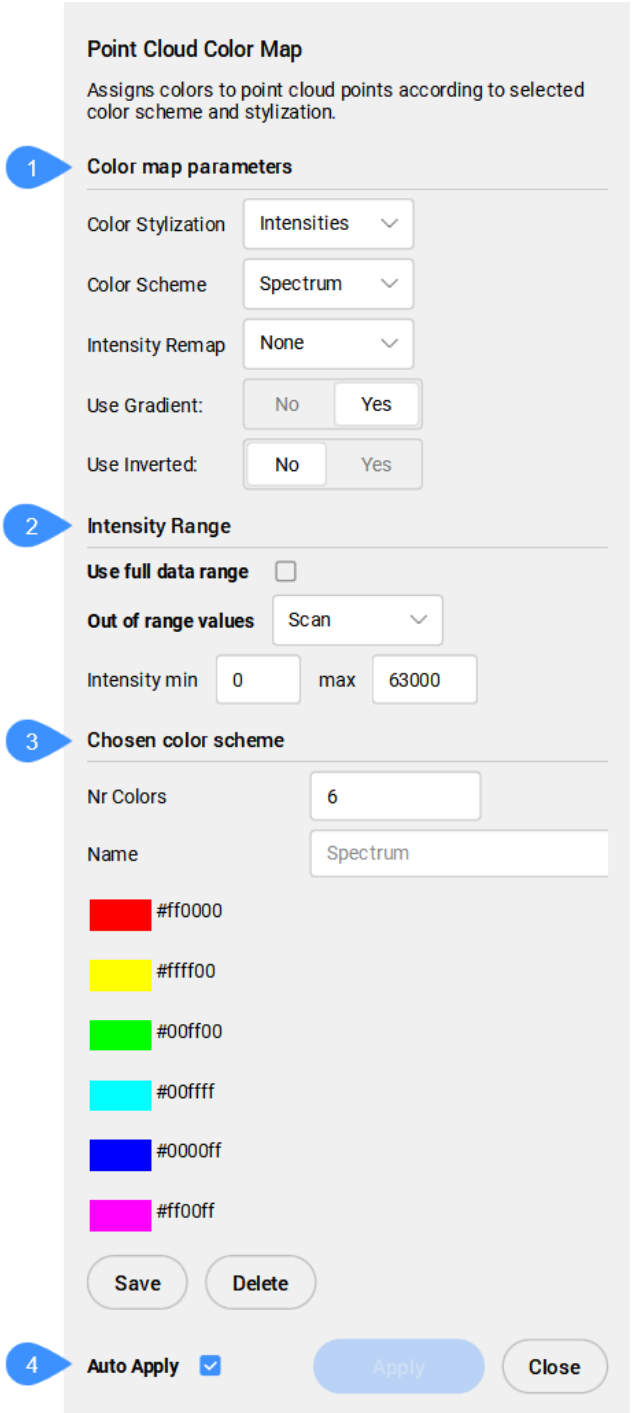
Colorizes the point cloud based on a range of colors. You can define the options through the **Point Cloud Color Map** command context panel, as well as through the Command line.

21.60.2 Method

The **Point Cloud Color Map** command context panel is automatically displayed when running the command. You can specify color map parameters and color schemes.

21.60.3 Options within the command context panel

Note: The options within the command context panel reflect the options within the Command line.



1 Color map parameters

2 Intensity Range

3 Chosen color scheme

4 Auto Apply

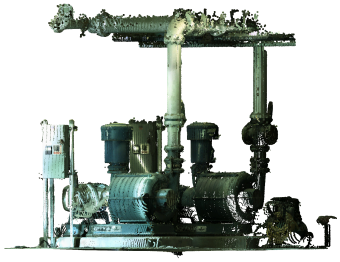
Color map parameters

Allows you to set the color map parameters.

Color Stylization

Specifies the color stylization. Depending on the chosen color stylization, different setting are further available.

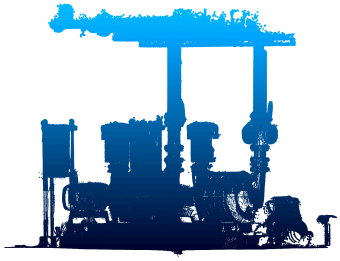
Scan: assigns colors based on the color data collected by the scan.



Object: uses the color assigned to the point cloud entity in the DWG file. This color can be retrieved and changed via the **Properties** panel with the point cloud selected.

Elevation: assigns colors based on the elevation of the points using the specified color map.

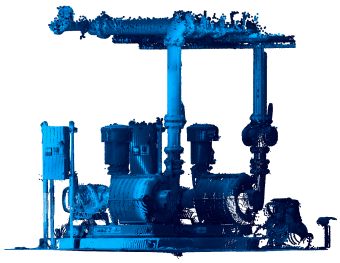
The predefined color schemes for this color stylization are **Hydro**, **Grayscale**, **Earth**, **Blues**, **Greens**, **Reds**, and **Spectrum**.



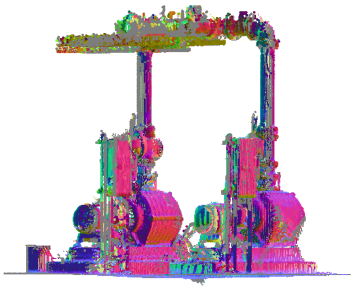
Intensities: assigns colors according to their intensity value using the specified color map.

> ***Note:*** When the point cloud file does not contain any intensity data, the colors are assigned by height, regardless of the option you select.

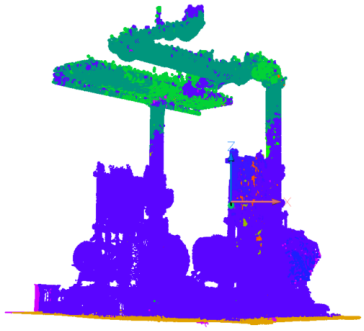
The predefined color schemes for this color stylization are **Hydro**, **Grayscale**, **Earth**, **Blues**, **Greens**, **Reds**, and **Spectrum**.



Normals: assigns colors to points based on their normals using the specified color map.

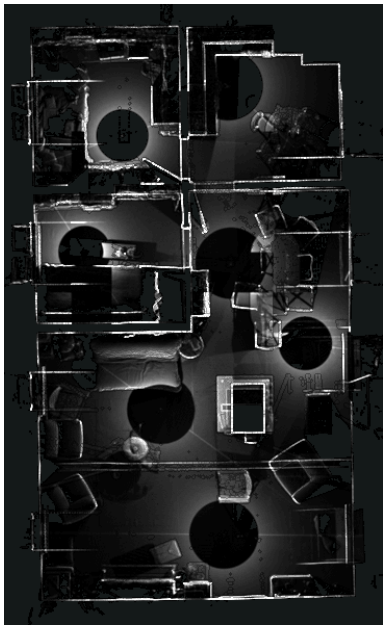


Classification: assigns colors to points based on their classification using the specified color map.
The predefined color schemes for this color stylization are **Lidar Classifications** and **3DR Classifications**.



Note: If the points are not classified, use the POINTCLOUDCLASSIFY command before using the POINTCLOUDCOLORMAP command.

X-Ray: adds transparency at the point cloud entity level.
Allows you to trace a point cloud and create 2D drawings based on orthographic views. See also the **SECTIONPLANE command** and **ACTIVATESECTIONVIEW command** articles.



Note: For Linux, force BricsCAD to run on the GPU for this feature to perform well.

Color Scheme

Hydro: assigns a range of colors from deep sky blue to very dark blue.

Grayscale: assigns a range of colors from light gray to dark gray.

Earth: assigns a range of colors from light brown to dark brown.

Blues: assigns a range of colors from blue to very dark blue.

Greens: assigns a range of colors from light green to dark green.

Reds: assigns a range of colors from light red to dark red.

Spectrum: assigns a range of colors based across the color spectrum.

Intensity Remap

By default this value is set to **None**. In order to improve the intensity contrast in the darker areas, select one of the two functions to be applied to the intensities first:

Gamma is a power function (the square root) of the input intensities.

Shadows is a function applied to input intensities, also to get better contrast in darker areas before mapping them to color.

Use Gradient

Specifies whether to use a gradient or not:

No: no gradient is applied, the colorization of the point cloud is defined by using value intervals defined by the number of colors used in the scheme. The desired number of colors needs to be specified.

Yes: a gradient will be applied to the point cloud points.

Use Inverted

Specifies whether the order of the colors in the selected color scheme should be used inverted.

No: the order of the colors specified in the selected color scheme is used.

Yes: the inverted order of the colors specified in the selected color scheme is used.

Intensity Range

Determines the min and max values for scaling **Intensity** and **Elevation** used in colormaps.

Use full data range

When turned **On**, all visible points are taken into account to determine the min and max values. Turn **Off** to manually set these values.

Out of range values

Specifies the colors assigned to points found out of the specified range.

Scan

Assigns the original RGB color to points found out of the specified range.

MinMax

Assigns the min and max colors to points found below or above the specified range.

Intensity min

Sets the intensity minimum value.

Intensity max

Sets the intensity maximum value.

Chosen color scheme

Allows you to edit an existing **Color Scheme** or to create a new one. These maps are stored in the DWG file in an AutoCAD® compatible way. Click on color listed in the selected **Color Scheme** to open the **Choose a color** dialog box that lets you select a new color.

Number of colors
Specifies the number of colors used in the **Color Scheme**. Enter a value to change this number.

Name
Specifies a name for the new custom **Color Scheme**.

Save
Saves the modifications made to the color scheme. If no name is specified in the **Name** field, the current **Color Scheme** is updated.

Delete
Deletes the selected custom **Color Scheme**.

Auto Apply
When checked, modifications are automatically applied to the point cloud.

Note: When this option is not checked, changes can be applied manually by pressing **Apply**.

21.61 POINTCLOUDCROP command

Creates a crop boundary on a point cloud.

Shape Lite Pro Mechanical BIM

Icons:

21.61.1 Description

Creates a crop boundary on an attached point cloud to limit the displayed points to a specific area. If the current drawing has only one point cloud attached, it is automatically selected for the crop operation. If the current drawing has more than one point cloud attached, you are prompted so select which point cloud to crop.

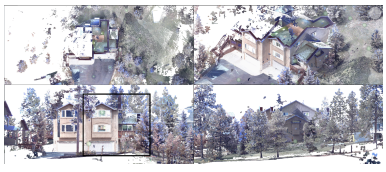
21.61.2 Method

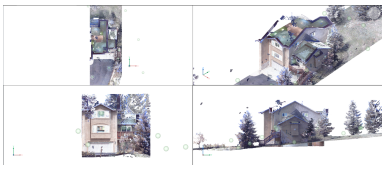
Crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume.

21.61.3 Options within the command

Rectangular
Creates a rectangular prismatic cropping.

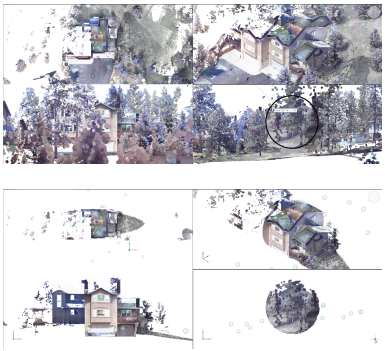
Note: The crop boundary creates a rectangular volume of points perpendicular to the view in which you defined it.





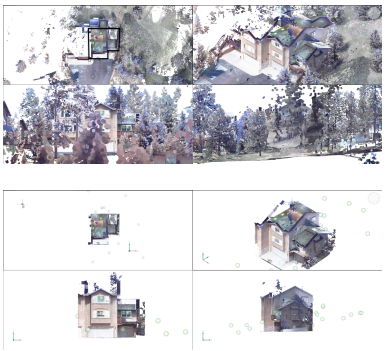
Circular
Creates a cylindrical cropping

Note: The crop boundary creates a circular volume of points perpendicular to the view in which you defined it.

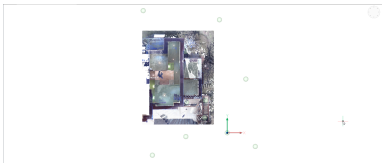


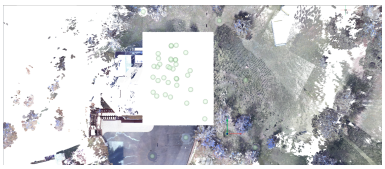
Polygonal
Creates a polygonal prismatic cropping.

Note: The crop boundary creates a polygonal volume of points perpendicular to the view in which you defined it.

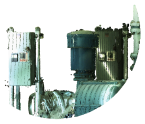


Invert
Inverts the current cropping. Toggles between Inside/Outside.

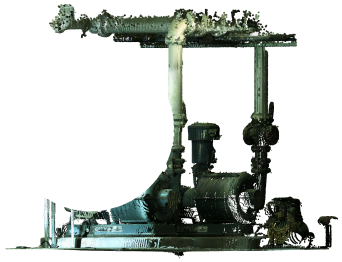




Inside
Displays the part of the point cloud inside the volume only.



Outside
Displays the part of the point cloud outside the volume only.



showToggle
Toggles the current cropping on/off.

removeLast
Removes the most recently added cropping volume.

removeAll
Removes all cropping.

21.62 POINTCLOUDCROPSOLID command

Converts an extruded solid/polysolid to a crop solid.

Shape Lite Pro Mechanical BIM

Icon:

Note: Crop solids can be used to crop point clouds in 3D and can be modified like any other BricsCAD solid.

21.62.1 Method

There are two methods to convert solids or polysolids to a crop solid:

- Inside

- Outside

21.62.2 Options within the command

Inside
Converts the selected solids into inside crop solids.

Outside
Converts the selected solids into outside crop solids.

Note: You can switch the Inside/Outside cropping property of a crop solid from the **Properties** panel.

21.63 POINTCLOUDDELETEITEM command

Deletes point cloud files.

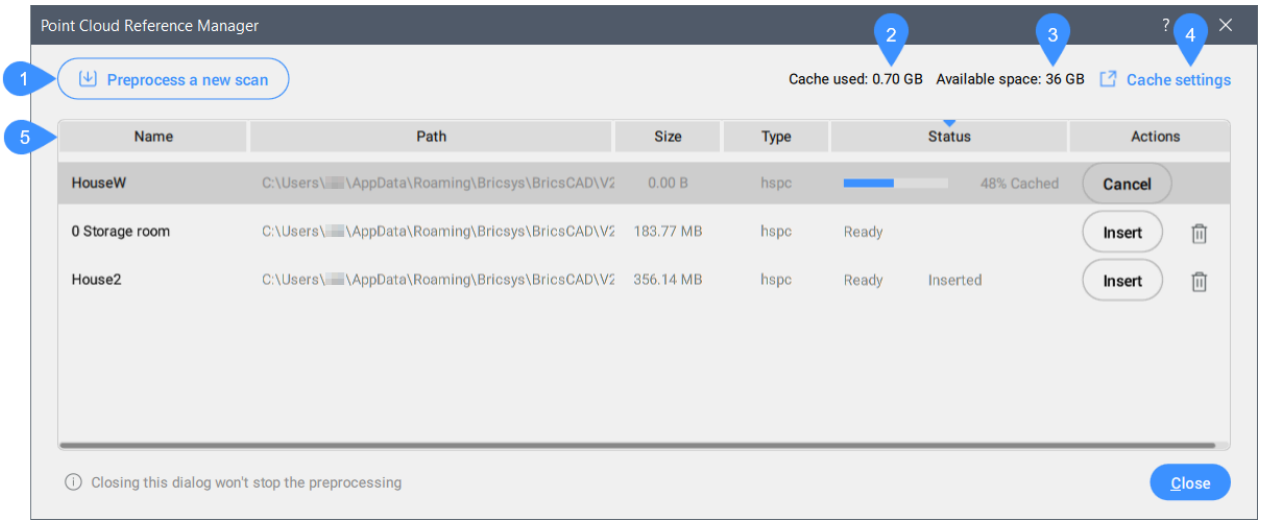
Shape Lite Pro Mechanical BIM

Icon:

21.63.1 Description

Opens the **Point Cloud Reference Manager** dialog box to delete point cloud files from the cache.

Note: It also removes all the insertions (if any) of that point cloud in the current drawing.



1 Preprocess a new scan

2 Cache used

3 Available space

4 Cache settings

5 Table of scans

21.63.2 Preprocess a new scan

Allows you to select a new point cloud file/folder to preprocess and opens the **Attach Point Cloud** dialog box. The **In progress bar** is displayed in the **Status** field of the **Point Cloud Reference Manager**dialog box while processing. A notification balloon displays when the point cloud is fully cached.

Note: When geographic information is available in the original data, it is stored in the point cloud's cache (only if the point cloud is preprocessed in HSPC format). You can access this information in the **Properties** panel under the **Misc** category for a selected point cloud instance. To overwrite the DWG file's geographical information with that of the point cloud instance (possible only if the CRS of the point cloud is the same as the CRS of the DWG file), turn On the **Geolocate** property of the selected point cloud instance.

21.63.3 Cache used

Specifies the total amount of the used cache.

21.63.4 Available space

Displays the total amount of available cache.

21.63.5 Cache folder settings

Opens the **Settings** dialog box to set the POINTCLOUDCACHEFOLDER system variable value that specifies the folder(s) where point cloud cache files are stored.

21.63.6 Table of scans

Lists the available preprocessed point cloud scans.

Right click a column header to display the table context menu:

- **Columns list**: Show/hide a table columns by clicking on its name in the list.
- **Show all columns**: Shows all table's columns.
- **Restore column positions**: Restores the default position of all table's columns.

Table columns
You can show or hide columns by right clicking on a column header and selecting column names in the context menu.

Name
Displays the point cloud file name.

Note:

- The point cloud cache name is a user specified name for the preprocessed point cloud. It's the default point cloud name if you do not specify a point cloud name for insertion.
- The double-byte characters are accepted for the point cloud name.

Path
Displays the point cloud file location.

Size
Displays the point cloud file size in MB.

Status
Displays the point cloud file status:

- **In progress bar**: the file is in preprocessing.
- **Ready**: available for **Insert** or **Delete**.
- **Inserted**: the file is inserted in the current drawing.

Action area

Cancel
Allows you to interrupt the preprocessing of the file. Once the point cloud file is preprocessed and ready to use, the **Cancel** button disappears.

Insert
Opens the **Attach Point Cloud** dialog box and attaches the point cloud file in the current drawing.

Note: Multiple insertions can have the same or a different point cloud name.

Delete
Removes the point cloud file from the cache.

Note: It also removes all the insertions (if any) of that point cloud in the current drawing.

Type
Displays to which type the inserted file has been preprocessed. If the POINTCLOUDHSPC system variable is ON, the file is inserted as an HSPC type.

21.64 -POINTCLOUDDELETEITEM command

Deletes point cloud files.

Shape Lite Pro Mechanical BIM

21.64.1 Description

Deletes point cloud files from the cache through the Command line.

Note: It also removes all the insertions (if any) of that point cloud in the current drawing.

21.64.2 Method

Enter the index of the point cloud to delete it.

Note:

- You can separate indexes by commas to delete multiple point clouds.
- Type **All** to delete all.
- The cached point clouds are listed in the Command line history.

21.65 POINTCLOUDDETECTFLOORS command

Assigns selected points to detected floors.

Shape Lite Pro Mechanical BIM

Icon: 

21.65.1 Description

Detects floors and ceilings in a point cloud representing a building and assigns points to the detected floors. The detection is based on regions of points with similar Z- coordinates.

Note: The POINTCLOUDDETECTFLOORS command is used as a step in the point cloud Scan to BIM workflow. See the article **Point cloud Scan to BIM workflow**.

The detected floors are listed in the **Point Cloud Manager** panel under **Buildings**.

Note:

- To edit the selection of points assigned to a floor, right-click the floor listed in the **Point Cloud Manager** panel and select the **Modify** option from the context-menu (see the **POINTCLOUDMODIFYFLOOR command** article).
- To manually create a floor from scratch, right-click the building listed in the **Point Cloud Manager** panel and select the **Add Floor** option from the context-menu (or use the **Add** option of the POINTCLOUDMODIFYFLOOR command).

21.65.2 Options within the command

Full
Automatically detects floors for the entire point cloud.

Selection
Specify the point cloud area where to detect floors.

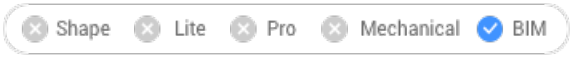
Create spatial elements
Choose **Yes** to let the command create spatial elements based on the detected floors height.

Note:

- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.
- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

21.66 POINTCLOUDDETECTROOMS command

Assigns each point to a room.



Icon: 

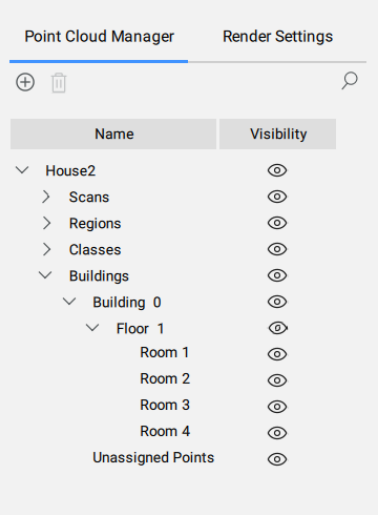
21.66.1 Method

The command searches for enclosed spaces in a point cloud that could form rooms and classifies them under different rooms.

The command can detect rooms for all the floors listed in the **Point Cloud Manager** panel or for one floor with the specified Id.

Note: The POINTCLOUDDETECTROOMS command is used as a step in the point cloud Scan to BIM workflow (see the article **Point cloud Scan to BIM workflow**).

The detected rooms are listed in the **Point Cloud Manager** panel under **Buildings**.



Access different options from the **Point Cloud Manager** panel's context menus. Select items listed in the panel and right-click the selection:

Room

Edit

Allows you to edit the contour of the selected room.
Enables the **Top** view and lets you reposition the vertices of the closed polyline.
To exit the editing mode, right-click the room item again and select **Accept** or **Decline**.

Rename

Allows you to type in a new room name.

Note:

- This option can also be accessed by double-clicking a selected room's name.
- The room keeps the number assigned during the detection process.

Multiple rooms

Merge rooms

The points corresponding to the selected rooms are classified under a single room.

Floor

Add Room

Allows you to manually add a new room by selecting a point cloud area.

Enables the **Top** view and lets you define a closed polyline.

Note:

- The **Toggle Visibility** option, available in each item's context menu, allows you to switch On/Off the visibility of the selected item.
- Where available, the **Delete** option deletes the selected item/items from the list and removes the specific classification of the corresponding points.

Note: Deleting the **Buildings** category from the **Point Cloud Manager** panel removes all building-related information (building, floor, room) for the items listed in this category.

21.67 POINTCLOUDDEVIATION command

Identifies and visually transposes the closeness of fit of point cloud points with respect to their closest selected planar (sub)entity or TIN surface.

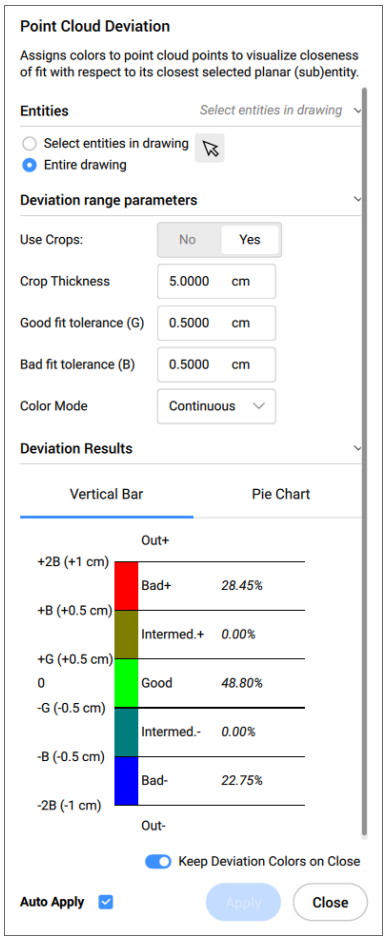
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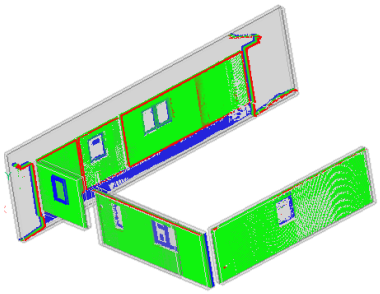
Icon: 

21.67.1 Method

Assigns colors to point cloud points to visualize their closeness of fit with respect to the closest selected planar (sub)entity or TIN surface. The selected entities are isolated using crops while the **Command Context** panel is open.

After selecting the planar surface, TIN surface or the solid to compare the point cloud to, the **Point Cloud Deviation** Command Context panel opens to let you easily change the settings.





21.67.2 Options within the command context panel

Entities

Select entities in the drawing

Selects specific geometric entities in the drawing.

Entire drawing

Selects all geometric entities in the drawing.

Deviation range parameters

Note: Parameters previously set by the user are remembered.

Use Crops
Toggles the crop of the solids.

Note: This option is available only when the input entity is a solid.

Crop Thickness
Defines the thickness of a solid that is used around the geometry as a reference for proximity analysis.

Note: This option is available only when the input entity is a solid.

Good fit tolerance (G)
Sets the good fit (OK) tolerance value.

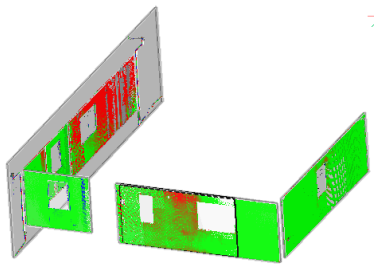
Bad fit tolerance (B):
Sets the bad fit (warning) tolerance value.

Color Mode

Continuous
Spreads colors continuously

Discrete
Spreads colors discontinuously.

Deviation Results
Displays the point cloud distribution in percentage and colors. You can switch between the **Vertical Bar** and **Pie Chart** tabs to visualize the percentages.



Keep Deviation Colors on Close
Toggles the visualisation of the point cloud distribution in colors after closing the panel.

Auto Apply
Automatically applies the changes made in the **Command Context** panel.

Note: The options within the **Command Context** panel reflect the options within the Command line.

21.68 POINTCLOUDDOLLHOUSE command

Hides all faces with normals in the viewing direction.

Shape Lite Pro Mechanical BIM

Icon:

21.68.1 Description

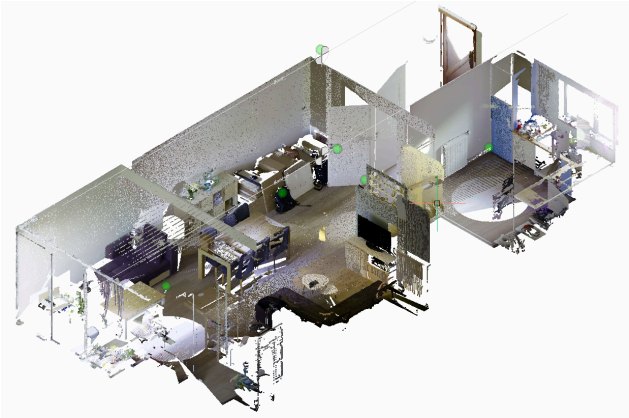
This feature allows easier viewing of an interior scan by hiding the points whose normal vectors point towards the camera.

Note: The command is supported only for point clouds for which normal vectors information is available.

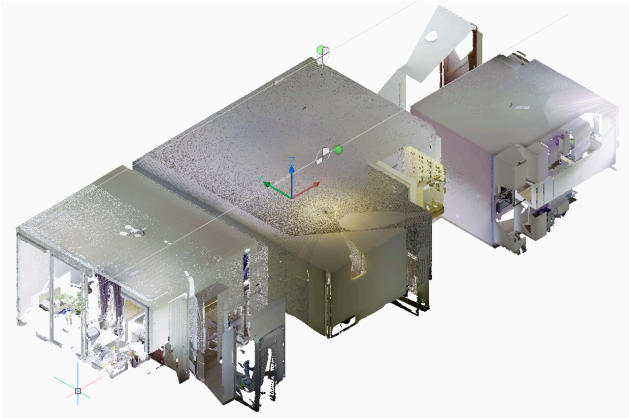
- For structured point cloud data, normal vectors are calculated during point cloud preprocessing. In case a point cloud has been preprocessed on an older BricsCAD version where normals have not yet been calculated, use the POINTCLOUDNORMALS command first.
- Normal vectors calculation requires a point cloud processed in HSPC format. Both POINTCLOUDHSPC and POINTCLOUDNORMALS system variables need to be set to On (the default value for both system variable is On).

21.68.2 Options within the command

Yes
Hides points with normals in the view direction.



No
Does not hide points with normals in the view direction.



21.69 POINTCLOUDEXPORT command

Exports the visible parts of a point cloud to a .pts, .Hspc or a .laz file.

Shape Lite Pro Mechanical BIM

Icon:

21.69.1 Description

Saves data from the current drawing to the specified file format [Pts/Hspc/Laz].

Note: The HSPC file format is a proprietary format developed by Hexagon VCH (Visual Computing Hub).

21.70 -POINTCLOUDEXPORT command

Exports the visible parts of a point cloud via Command line.

Shape Lite Pro Mechanical BIM

21.70.1 Method

Specify the export format. Then, insert the full path, file name, and the corresponding extension for the exported point cloud file.

21.70.2 Options within the command

Specify the export format
Allows you to choose the export format.

Pts
Exports the point cloud file to PTS format.

Laz
Exports the point cloud file to LAZ format.

21.71 POINTCLOUDFITCYLINDER command

Creates cylinders in Point Clouds.

Shape Lite Pro Mechanical BIM

Icon:

21.71.1 Description

This command can only operate in **Bubble Viewer**.

21.71.2 Method

Open **Bubble Viewer** and choose two points along the axis to create a cylinder.

After selecting the second point, the cylinder is inserted in model space. Press **Ctrl+A** keys to sync the drawing view to match the **Bubble Viewer** and see the new inserted cylinder.

21.71.3 Options within the command

Accept
Accepts the new inserted cylinder.

Decline
Does not accept the new inserted cylinder.

21.72 POINTCLOUDFITPLANAR command

Creates a planar surface or a solid from a point cloud point.

Shape Lite Pro Mechanical BIM

Icon:

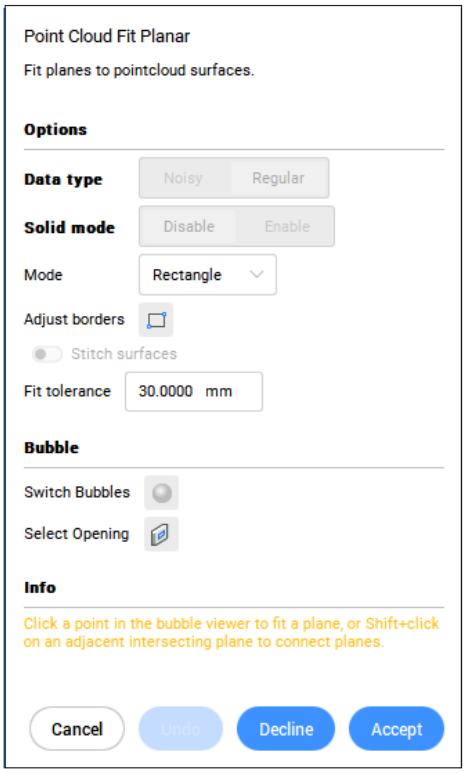
21.72.1 Method

Searches a point cloud for a dense set of planar points around selected seed point(s) and creates a planar surface or a solid if possible.

Note: Enables to create multiple surfaces or solids until you cancel.

Note: You can execute the command in model view or in a bubble view. Double click the bubble in which you want to start the command before launching the command.

This command opens the **Point Cloud Fit Planar** command panel.



21.72.2 Options within the command

Select point cloud entity
Specifies point cloud entity.

Data type

- **Noisy**: robust detection for coarse/noisy data - requires selecting 3 seed points.
- **Regular**: normal data - requires selecting 1 seed point.

Solid mode
Searches for a parallel plain in the point cloud to create a solid. For example a wall or a slab.

- **Disable**: disables the computation of solid mode data in order to save time.
- **Enable**: enables the computation of solid mode data.

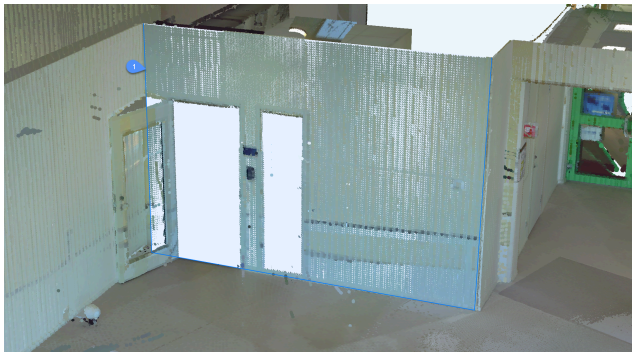
Note: Solid mode is available in Model view only, if a dense area is detected in a plane that is parallel to the detected surface.

Select seed point in model view
Specifies the seed point for creating a planar surface or solid.

Mode
Switches between:

- **rectangle** (1): creates a single plane around the chosen point, rectangle outer border;
- **contour** (2): creates a single plane around the chosen point, contoured outer border;
- **solid** (3): creates a cuboid solid with a face that contains the specified seed point and an automatically detected second plane nearby.

Note: Press **Ctrl** to switch between the available modes.







adJust borders
Defines a new border for the selected plane by specifying the vertices (Select vertices on border).

Stitch surfaces
Stitches adjacent surfaces selected in the rapid selection mode.

Note: Rapid selection is activated by holding down the **Shift** key.

Fit tolerances
Sets a tolerance value for the plane fit.

Accept
Accepts the current planar surface or solid.

Decline
Declines the current planar surface or solid.

Undo
Removes the most recently added surface from the current selection set. Use **Decline** option to remove the complete set.

Bubble



Note: Use **Shift + click** key for rapid selection of adjacent surfaces. If **Stitch surfaces** is on, these surfaces will be stitched together.

Switch Bubbles
Switches between bubble viewers. Selects the bubble to switch to. Only available in bubble mode.

Select Opening
Creates an opening by selecting a point on a opening in a surface. Only available in bubble mode.

21.73 POINTCLOUDFITROOMS command

Creates solids corresponding to detected rooms in a point cloud.

Shape Lite Pro Mechanical BIM

Icon: 

21.73.1 Method

The command creates solids corresponding to detected rooms in a point cloud.

The command is supported only for point clouds for which normal vectors information is available.

For structured point cloud data, normal vectors are calculated during point cloud preprocessing. In case a point cloud has been preprocessed on an older BricsCAD version where normals have not yet been calculated, use the POINTCLOUDNORMALS command first.

Note: Normal vectors calculation requires a point cloud processed in HSPC format. Both POINTCLOUDHSPC and POINTCLOUDNORMALS system variable need to be set to On (On by default).

For unstructured point cloud data, normal vectors information is imported if present and used by the command. Otherwise it is not calculated and the command is not supported.

Note:

- The POINTCLOUDFITROOMS command is used as a step in the point cloud Scan to BIM workflow. For more information, see the article **Point cloud Scan to BIM workflow**.
- You can continue working during the background processing.

21.73.2 Options within the command

All
Creates solids for all detected rooms.

Id
Creates a solid for the room with the specified Id.

Restrict detection to perpendicular (90°) walls
Select **Yes** if the rooms have two perpendicular wall directions.
Select **No** if the rooms have wall directions that are not perpendicular.

Note: The POINTCLOUDFITROOMS command is a Beta feature and it will be further developed. Backward compatibility is not guaranteed.

21.74 POINTCLOUDGEOGRAPHICLOCATION command

Sets the geographic location based on point cloud.

Shape Lite Pro Mechanical BIM

Sets the geographic location based on point cloud; sets point cloud transform based on geographic location; relatively aligns two point clouds.

Note: Only LAS and LAZ file formats can make use of this functionality.

21.74.1 Method

There are three methods:

- Geolocation
- Transform
- Relative

21.74.2 Options within the command

Geolocation
Allows to set a geographic marker in the DWG based on a point cloud with geographic location.

Transform
Updates the point cloud transform based on a geographic location in the DWG, as well as the geographic information in the point cloud.

Relative
Relatively aligns two point clouds if both contain a geographic location.

21.75 POINTCLOUDMANAGERPANELCLOSE command

Closes the **Point Cloud Manager** panel.

Shape Lite Pro Mechanical BIM

21.75.1 Description

Closes the **Point Cloud Manager** panel to hide it from the current workspace. If the **Point Cloud Manager** panel is stacked when you close it, the **Point Cloud Manager** tab or icon is removed from the stack.

21.76 POINTCLOUDMANAGERPANELOPEN command

Opens the **Point Cloud Manager** panel.

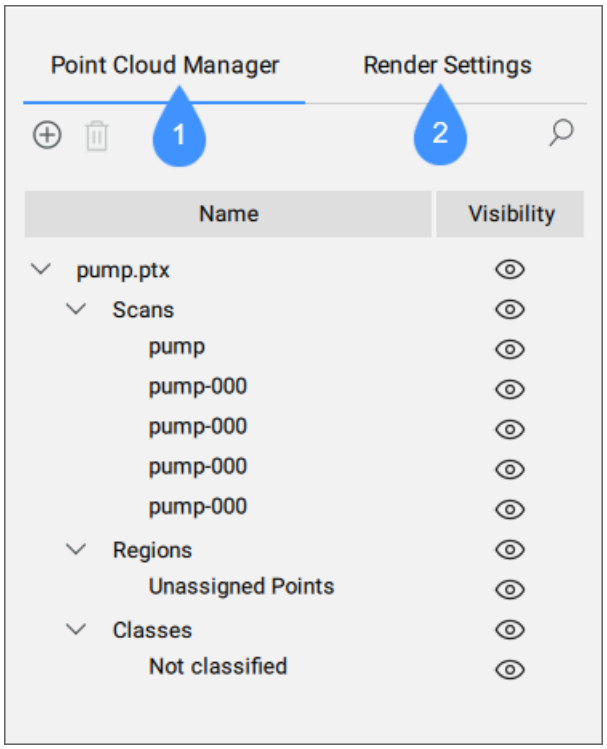
Shape Lite Pro Mechanical BIM

21.76.1 Description

Opens the **Point Cloud Manager** panel to display it in the current workspace. The **Point Cloud Manager** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Point Cloud Manager** panel can be either floating, docked or stacked.

The **Point Cloud Manager** panel creates and manages regions in a point cloud and controls the display of point clouds based on scans/waypoints, classification and regions.

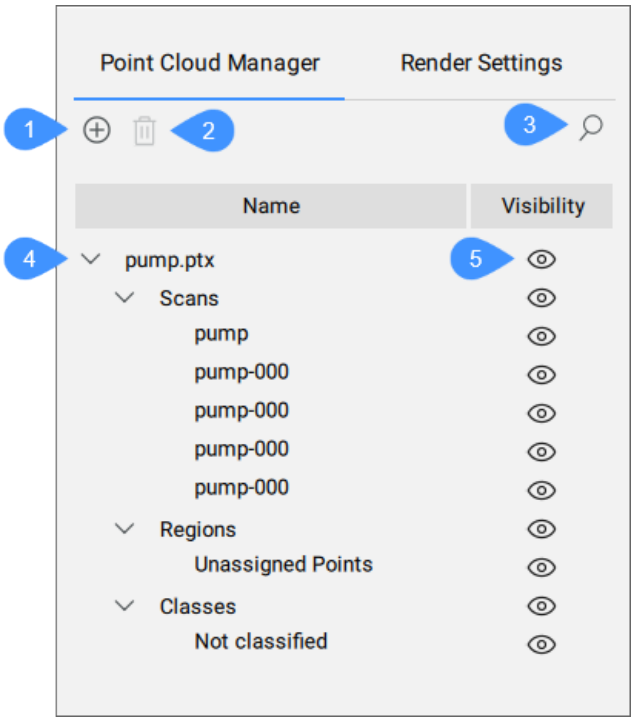
Note: Scan information will only be available if the point cloud is preprocessed in HSPC format. Only structured point clouds contain scan information. Classification information will be available if the point cloud data source contains classification information (some *.LAS/*.LAZ files) and it is preprocessed in HSPC, or if the point cloud is classified using the Hexagon point cloud classifier (POINTCLOUDCLASSIFY command).



1 Point Cloud Manager tab

2 Render Settings tab

21.76.2 Point Cloud Manager tab



1 New region

2 Remove region

3 Search

4 List of point clouds and their scans, regions, classes, and building information

5 Visibility

New region
Allows you to create a region by running the POINTCLOUDREGION command:

Draw region
Runs the **Draw** option of the POINTCLOUDREGION command.

Convert crop solids to region
Runs the **Convert** option of the POINTCLOUDREGION command.

Remove region
Removes the selected region.

Note: The **Unassigned Points** region cannot be removed.

Search
Allows you to search for a specific scan, class or region name. The list is shortened and only shows the items that contain the entered sequence of characters, regardless of where they are located in the item's name.

List of point clouds and their scans, regions, classes, and building information
Displays a tree view list of point clouds with their scans, regions, and classes. Building information (building/floor/room) is also displayed after the POINTCLOUDDETECTFLOORS and POINTCLOUDDETECTROOMS commands (BricsCAD BIM only) have been used.
Right-click an item in the list to open a context menu:

Note:

- The **Toggle Visibility** option, available in each item's context menu, allows you to switch On/Off the visibility of the selected item.
- Where available, the **Delete** option deletes the selected item/items from the list and removes the specific classification of the corresponding points.

Note: Deleting the **Buildings** category from the **Point Cloud Manager** panel removes all building-related information (building, floor, room) for the items listed in this category.

Scans

Rename
Allows you to type in a new name.

Note: Double click an item to activate the rename mode.

Open scan in bubble viewer
Opens the **Bubble viewer** displaying the selected scan.
Nearby scans are indicated in the viewer by red bubbles. Hover the cursor over a bubble to display the scan name. Double-click a bubble to open the scan in the viewer.

Note: Hover over scan items in the list to show a preview of the scan.

Regions

Rename

Allows you to type in a new name.

Note: Double click an item to activate the rename mode.

Classes

Add new class

Allows you to add a new class to the **Classes** list by selecting one from the predefined lists: **Lidar**, **Indoor**, **Construction**, or add a custom class using the **Custom...** option.

Note: Use the POINTCLOUDCLASSIFY command to automatically classify points.

Add points

Runs the POINTCLOUDMODIFYCLASS command.

Allows you to manually classify points under the selected class by selecting points in the point cloud. Two selection methods are available: **Rectangular** and **Polygonal**.

Select

Allows you to add more points to the selection set.

Finish

Finishes the selection process and classifies the points.

Note: Use the POINTCLOUDCOLORMAP command and select **Classification** for **Color Stylization** to better visualize how the points are classified.

Buildings

Add Building

Runs the **Add** option of the POINTCLOUDMODIFYBUILDING command.

Allows you to specify the area of points to be classified under a new building.

Building

Right-click a building item to display a context menu:

Modify

Runs the **Edit** option of the POINTCLOUDMODIFYBUILDING command.

Allows you to modify the volume of the building (points assigned) by moving the faces of the temporary crop solid.

Add Floor

Runs the **Add** option of the POINTCLOUDMODIFYFLOOR command.

Allows you to specify the elevation and height of the new floor.

Building floors

Right-click a floor item to display a context menu:

Modify

Runs the **Edit** option of the POINTCLOUDMODIFYFLOOR command.

Allows you to modify the volume of the floor (points assigned) by moving the faces of the temporary crop solid.

Detect Rooms

Automatically detects rooms in the selected floor (see POINTCLOUDDETECTROOMS command).

Add Room
Allows you to manually add a new room by selecting a point cloud area.
Enables the **Top** view and lets you define a closed polyline.

Building rooms
Right-click a room item to display a context menu:

Edit
Allows you to edit the contour of the selected room.
Enables the **Top** view and lets you reposition the vertices of the closed polyline.
To exit the editing mode, right-click the room item again and select **Accept** or **Decline**.

Rename
Allows you to type in a new room name.

Note:

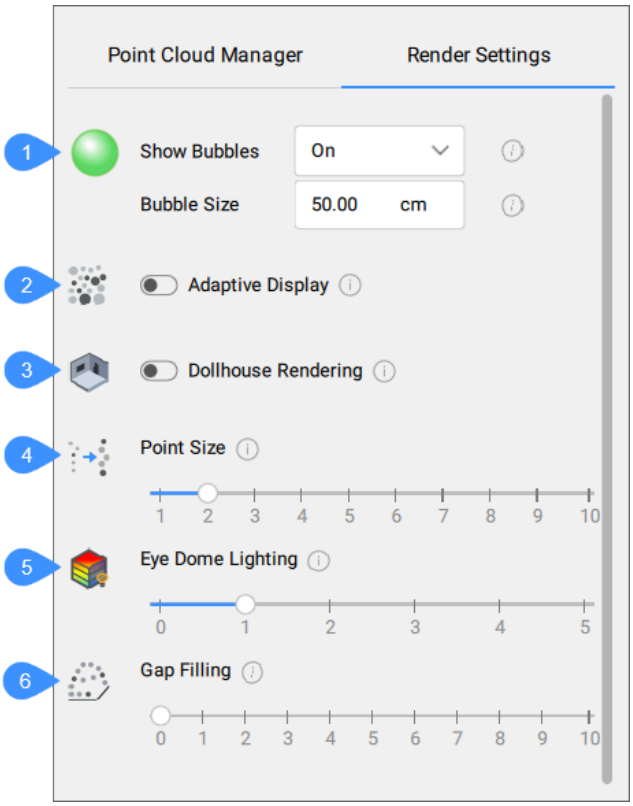
- This option can also be accessed by double-clicking a selected room's name.
- The room keeps the number assigned during the detection process.

Multiple building rooms
Select multiple room items and right-click to display a context menu:

Merge rooms
The points corresponding to the selected rooms are classified under a single room.

Visibility
Displays the current visibility state of listed items. Click the icon to switch On/Off the visibility of the selected items.

21.76.3 Render Settings tab



1 Bubbles settings

2 Adaptive Display

3 Dollhouse Rendering

4 Point Size

5 Eye Dome Lighting

6 Gap Filling

Bubbles settings

Show Bubbles ( )
Allows you to set the visibility of scan bubbles in Model space and **Bubble viewer**:
Off: hides all bubbles in the point cloud.
On: displays all bubbles in the point cloud.
Visible Only: displays non-cropped bubbles only.

Note: This setting applies to all point clouds inserted in the drawing.

Bubbles Size
Allows you to set the size of scan bubbles in Model space.

Adaptive Display ( )
Toggles the adaptive display of point sizes.

Note:

- Available for Windows and Linux, only for structured point clouds.

- Sets the value of POINTCLOUDADAPTIVEDISPLAY system variable.

When enabled, point sizes are adjusted according to local point density and distance from the camera to the position of the scanned points.

Dollhouse Rendering ( )
Toggles the display of points with normals in the view direction.
When enabled, points with normals pointing away from the camera are not shown.

Note: Only works when normals are available.

Point Size ( )
Sets the size of points displayed.

Eye Dome Lighting ( )
Allows you to set the non-photo-realistic shading to improve depth perception.

Gap Filling ( )
Fills gaps between points on surfaces.

Note: Only works in perspective view, not in orthogonal view.

21.77 POINTCLOUDMODIFYBUILDING command

Adds a building or modifies existing ones.

Shape Lite Pro Mechanical BIM

21.77.1 Description

Adds a new building (spatial classification) to the **Buildings** list in the **Point Cloud Manager** panel or modifies an existing one.

21.77.2 Options within the command

Add
Adds a new building. Lets you select a point cloud area to assign the points to the new building.

Note: The **Add Building** option from the context menu of the **Buildings** category in the **Point Cloud Manager** panel runs the **Add** option of the POINTCLOUDMODIFYBUILDING command.

Edit
Modifies the volume of an existing building.
Specify the building index as listed in the **Point Cloud Manager** panel. Move the faces of the temporary crop solid.

Note: The **Modify** option from the context menu of a building listed in the **Point Cloud Manager** panel runs the **Edit** option of the POINTCLOUDMODIFYBUILDING command.

21.78 POINTCLOUDMODIFYCLASS command

Adds points to a point cloud class.

Shape Lite Pro Mechanical BIM

21.78.1 Description

Allows you to select points in a point cloud to assign to a different class.

Note: Use the POINTCLOUDCLASSIFY command to automatically classify points in a point cloud.

21.78.2 Method

After launching the command, the indexes of the available classes are listed in the upper field of the Command line.

Enter the index of the class under which the points will be classified.

Choose the points selection type, select the points, then pres **Enter** to confirm the selection.

21.78.3 Options within the command

Rectangular
Activates the rectangular selection type.

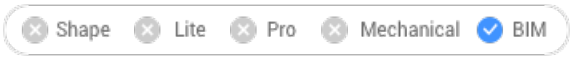
Polygonal
Activates the polygonal selection type.

Select
Allows you to add more points to the selection set.

Finish
Finishes the selection process and classifies the points.

21.79 POINTCLOUDMODIFYFLOOR command

Adds a floor or modifies existing ones.



21.79.1 Description

Adds a floor (spatial classification) to a building or modifies the volume of the specified floor by moving the faces of a temporary crop solid.

21.79.2 Options within the command

Add
Adds a new floor to an existing building. Specify the building index as listed in the **Point Cloud Manager** panel or **Spatial Locations Manager** dialog box.

Note: The **Add Floor** option from the context menu of a building listed in the **Point Cloud Manager** panel runs the **Add** option of the POINTCLOUDMODIFYFLOOR command.

Edit
Modifies the volume of a floor of an existing building. Specify indexes as listed in the **Point Cloud Manager** panel.
Move the faces of the temporary crop solid.

Note: The **Modify** option from the context menu of a floor listed in the **Point Cloud Manager** panel runs the **Edit** option of the POINTCLOUDMODIFYFLOOR command.

adjust Elevation
Adjusts the elevation of the temporary crop solid.

adjust Height
Adjusts the height of the temporary crop solid.

Done
Assigns the points in the temporary crop solid to the specified floor and exits the command.

21.80 POINTCLOUDNORMALS command

Calculates normals using the structured normal calculation after preprocessing.

Shape Lite Pro Mechanical BIM

21.80.1 Description

Calculates normals for structured point clouds (have bubbles) already available in the cache in HSPC that do not have normal vectors yet. The calculated normals will be written to the HSPC file.

Note: A pop-up message appears when the calculations are finished.

21.81 POINTCLOUDPOINTSIZE_MINUS command

Decreases the display size of points in a point cloud.

Shape Lite Pro Mechanical BIM

Icon: 

21.81.1 Description

Decreases the display size of points in a point cloud to see the individual scan points more clearly. This decreases the POINTCLOUDPOINTSIZE system variable by one.

This command displays no prompts and has no options.

21.82 POINTCLOUDPOINTSIZE_PLUS command

Increases the display size of points in a point cloud.

Shape Lite Pro Mechanical BIM

Icon: 

21.82.1 Description

Increases the display size of points in a point cloud to visually fill gaps between individual scan points. This increases the POINTCLOUDPOINTSIZE system variable by one.

This command displays no prompts and has no options.

21.83 POINTCLOUDPREPROCESS command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Note: Since BricsCAD V20, this command has been replaced by the POINTCLOUDATTACH command.

21.84 -POINTCLOUDPREPROCESS command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Note: Since BricsCAD V20, this command has been replaced by the -POINTCLOUDATTACH command.

21.85 POINTCLOUDPROJECTSECTION command

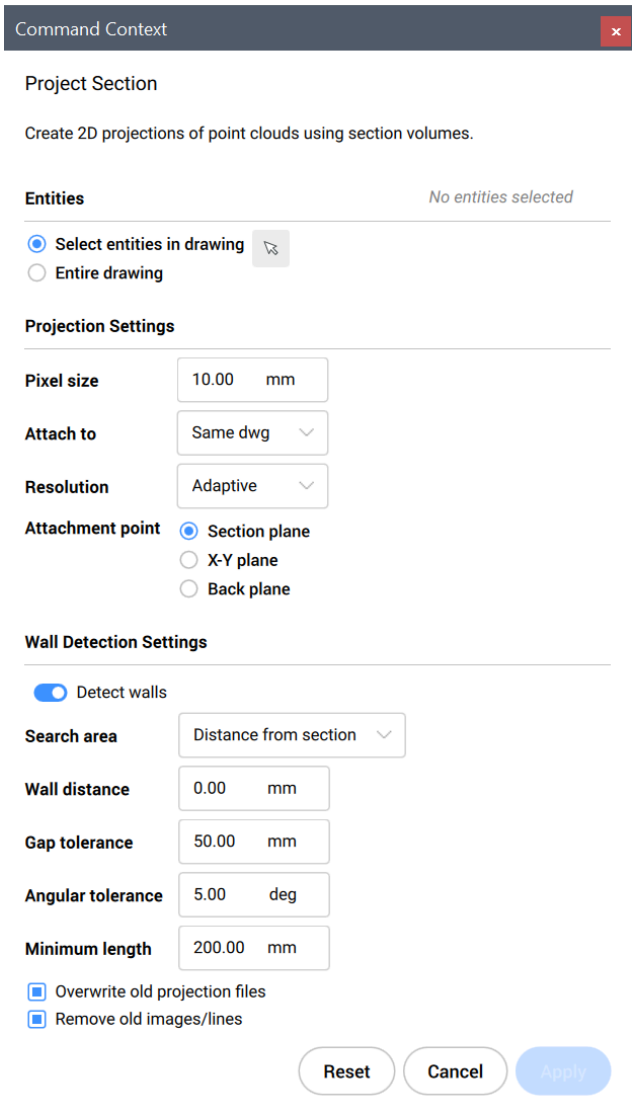
Automatically generates a 2D raster image with optional contour lines from a defined section box.

Shape Lite Pro Mechanical BIM

Icon: 

21.85.1 Description

Opens the **Command Context** panel.



Note: The options within the POINTCLOUDPROJECTSECTION command are identical to the ones in the **Command Context** panel.

21.85.2 Options within the command

change Selection
Allows you to select section volumes to project.

change Pixel size
Controls the pixel size of the generated image.

Attach to
Determines where the resulting image is generated. Allows you to switch between **Same.dwg** and **New destination**.

Same dwg
The projection is made in the same file.

Destination file
Creates a new .dwg file to make the projection.

Note: You can overwrite a previous file or create a new one.

Resolution
Determines the resolution of the projection. Allows you to switch between **Finest** and **Adaptive**.

Finest
Produces a sharp and detailed background picture. It uses all the points in the point cloud.

Note: The projection in the finest resolution would take longer to complete.

Adaptive
Produces a less detailed and softer background image.

aTtachment point
Determines the insertion point for the generated image. Allows you to switch between **Section plane**, **X-y plane** and **Back plane**.

Section plane
Attaches the projection to the section plane.

X-y plane
Attaches the projection flat to the X-Y plane.

Back plane
Attaches the projection to the back plane of the section volume.

detect Walls
Allows you to switch between **Yes** and **No**.

Yes
Lines representing walls will be generated with the raster image.

No
Only a raster image is created.

toLerances
Allows you to modify the parameters that control 2D line drawing.

Search area
Allows you to switch between **Across volume** and **Distance from section**.

Across volume
Detects all walls in the point clouds. This option is often used to generate floor plans.

Distance from section
Detects walls in a given radius from a plane. This option is often used to generate vertical sections.

Wall distance
Controls the search area for wall detection.

Gap
Controls the search area for the algorithm to draw lines. Gaps smaller than the specified value will be closed.

Angular
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.

Overwrite
Allows you to switch between **Yes** and **No**.
If applicable, it overwrites any old image files on disk related to this section volume.

reMove old
Allows you to switch between **Yes** and **No**.
If applicable, it removes previously added images or lines to this section volume.

21.86 POINTCLOUDREFERENCE command

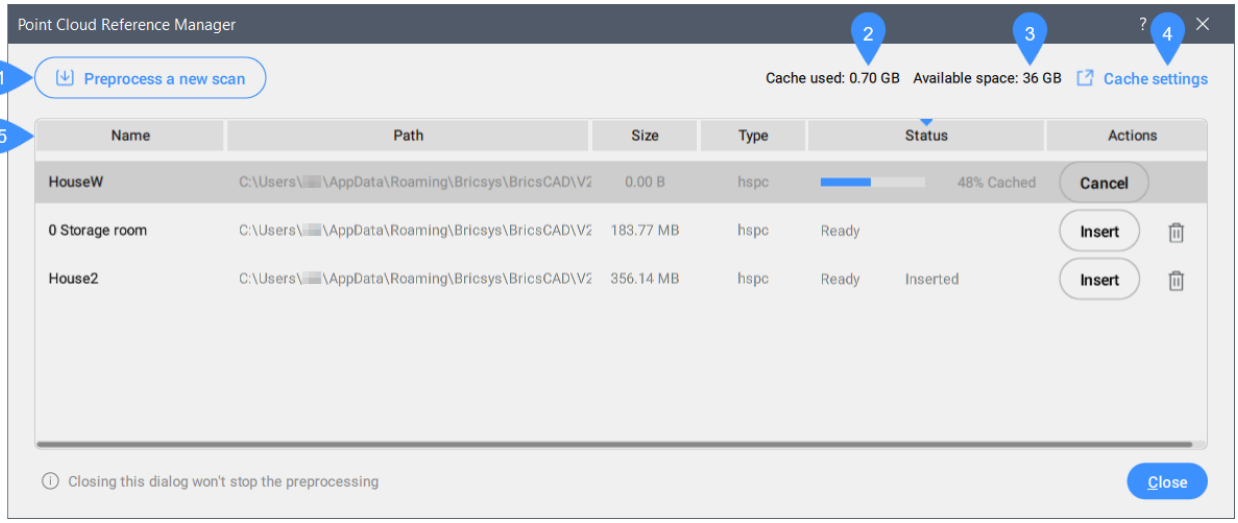
Loads point clouds from cache or inserts new point clouds to the current drawing.

Shape Lite Pro Mechanical BIM

Icon:

21.86.1 Description

Opens the **Point Cloud Reference Manager** dialog box where you can load point clouds from cache or insert new point clouds to the current drawing.



1 Preprocess a new scan

2 Cache used

3 Available space

4 Cache settings

5 Table of scans

21.86.2 Preprocess a new scan

Allows you to select a new point cloud file/folder to preprocess and opens the **Attach Point Cloud** dialog box. The **In progress bar** is displayed in the **Status** field of the **Point Cloud Reference Manager**dialog box while processing. A notification balloon displays when the point cloud is fully cached.

Note: When geographic information is available in the original data, it is stored in the point cloud's cache (only if the point cloud is preprocessed in HSPC format). You can access this information in the **Properties** panel under the **Misc** category for a selected point cloud instance. To overwrite the DWG file's geographical information with that of the point cloud instance (possible only if the CRS of the point cloud is the same as the CRS of the DWG file), turn On the **Geolocate** property of the selected point cloud instance.

21.86.3 Cache used

Specifies the total amount of the used cache.

21.86.4 Available space

Displays the total amount of available cache.

21.86.5 Cache folder settings

Opens the **Settings** dialog box to set the POINTCLOUDCACHEFOLDER system variable value that specifies the folder(s) where point cloud cache files are stored.

21.86.6 Table of scans

Lists the available preprocessed point cloud scans.

Right click a column header to display the table context menu:

- **Columns list**: Show/hide a table columns by clicking on its name in the list.
- **Show all columns**: Shows all table's columns.
- **Restore column positions**: Restores the default position of all table's columns.

Table columns
You can show or hide columns by right clicking on a column header and selecting column names in the context menu.

Name
Displays the point cloud file name.

Note:

- The point cloud cache name is a user specified name for the preprocessed point cloud. It's the default point cloud name if you do not specify a point cloud name for insertion.
- The double-byte characters are accepted for the point cloud name.

Path
Displays the point cloud file location.

Size
Displays the point cloud file size in MB.

Status
Displays the point cloud file status:

- **In progress bar**: the file is in preprocessing.
- **Ready**: available for **Insert** or **Delete**.
- **Inserted**: the file is inserted in the current drawing.

Action area

Cancel
Allows you to interrupt the preprocessing of the file. Once the point cloud file is preprocessed and ready to use, the **Cancel** button disappears.

Insert
Opens the **Attach Point Cloud** dialog box and attaches the point cloud file in the current drawing.

Note: Multiple insertions can have the same or a different point cloud name.

Delete
Removes the point cloud file from the cache.

Note: It also removes all the insertions (if any) of that point cloud in the current drawing.

Type
Displays to which type the inserted file has been preprocessed. If the POINTCLOUDHSPC system variable is ON, the file is inserted as an HSPC type.

21.87 -POINTCLOUDREFERENCE command

Loads point clouds from cache or inserts new point clouds into the current drawing via Command line.

Shape Lite Pro Mechanical BIM

21.87.1 Method

Import a new point cloud or load from cache.

21.87.2 Options within the command

Cache
Allows you to enter the index of the point cloud to attach it to the current drawing.

Note: The valid cached point clouds are listed in the Command line history.

New
Attaches a new point cloud to the current drawing.

Inputdata
Allows you to specify the point cloud link to attach to the current drawing.

Enter insertion point
Allows you to enter the insertion point coordinates or specify it on the screen.

Rotation angle
Allows you to enter the rotation angle or specify it on the screen.

Scale factor
Allows you to enter the scale factor or specify it on the screen.

21.88 POINTCLOUDREGION command

Creates regions in a point cloud.

Shape Lite Pro Mechanical BIM

21.88.1 Options within the command

Draw
Allows you to select parts of the point cloud to convert to a region.

Rectangular
Defines a rectangular region.

Polygonal
Defines a polygonal region.

Select
Allows you to continue selecting parts of the point cloud.

Finish
Finishes the selection process.

Convert
Converts solids or crop solids to regions.

Select solids or polysolids to convert
Allows you to select solids or polysolids to convert to a region.

Note: The solids or crop solids are not deleted after the region is created.

Some of the selected solids are not cropsolids, do you want to convert them?
Allows you to choose between **Yes** and **No**.

- **Yes**: converts the entities to crop solids.
- **No**: does not convert the entities to crop solids.

Select point cloud entity
If there are more point clouds attached in the drawing, it allows you to select the point cloud for which you can create regions.

Enter region name
Allows you to specify the name of the region.

Note: Press the **Esc** key to exit the command.

21.89 POINTCLOUDRENDERSETTINGSOPEN command

Opens the **Render Settings** tab in the **Point Cloud Manager** panel.

Shape Lite Pro Mechanical BIM

Icon: 

21.89.1 Description

Opens the **Point Cloud Manager** panel with the **Render Settings** tab opened. The **Point Cloud Manager** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Point Cloud Manager** panel can be either floating, docked or stacked.

21.90 POINTCLOUDSCAN2BIM command (Experimental)

Detects floors and rooms in a point cloud, then creates solids for detected rooms.

Shape Lite Pro Mechanical BIM

Icon: 

21.90.1 Disclaimer

EXP.

POINTCLOUDSCAN2BIM is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

21.90.2 Method

The command combines the POINTCLOUDDETECTFLOORS, POINTCLOUDDETECTROOMS, and POINTCLOUDFITROOMS commands in one step.

The command analyzes point clouds to detect floors and rooms. Then, it creates solids corresponding to detected rooms and gives the option to run the BIMINVERTSPACES command.

The command is supported only for point clouds for which normal vectors information is available.

For structured point cloud data, normal vectors are calculated during point cloud preprocessing. In case a point cloud has been preprocessed on an older BricsCAD version where normals have not yet been calculated, use the POINTCLOUDNORMALS command first.

Note: Normal vectors calculation requires a point cloud processed in HSPC format. Both POINTCLOUDHSPC and POINTCLOUDNORMALS system variable need to be set to On (On by default).

For unstructured point cloud data, normal vectors information is imported if present and used by the command. Otherwise it is not calculated and the command is not supported.

Note:

- You can continue working during the background processing.
- The point cloud Scan to BIM semi-automatic workflow is described in the **Point cloud Scan to BIM workflow** article.

21.90.3 Options within the command

Full
Analyzes the entire point cloud.

Selection
Allows you to specify the point cloud area to be analyzed.

Edit
Exits the command to let you edit the room solids.

Biminvertspaces
Runs the BIMINVERTSPACES command.

21.91 POINTCLOUDSHOWBUBBLES command

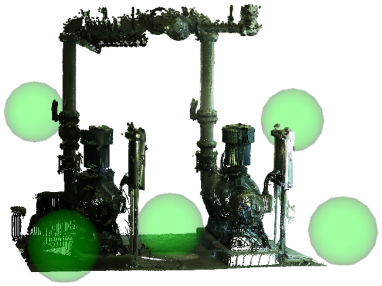
Toggles the display of point cloud bubbles.

Shape Lite Pro Mechanical BIM

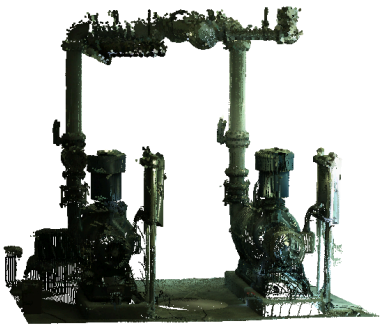
Note: You can change the bubbles size in the **Properties** panel.

21.91.1 Options within the command

Yes
Displays the bubbles.



No
Does not display the bubbles.



21.92 POINTCLOUDUNCROP command

Removes the crop display of a point cloud.

Shape Lite Pro Mechanical BIM

21.92.1 Description

Removes the crop display of a point cloud previously created with the POINTCLOUDCROP command.

21.93 POINTLIGHT command

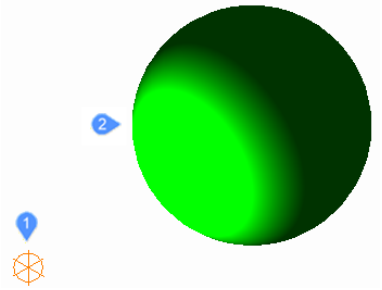
Places point lights.

Shape Lite Pro Mechanical BIM

Icon:

21.93.1 Description

Places point lights for use with renderings. A point light represents sources that illuminate in all directions, such as a bare lightbulb, and it so has no target.



1 Glyph for a point light illuminating in all directions.

2 Sphere illuminated by the point light.

Note: Unlike all other light types, point lights do not have a target or vector associated with them. Drawings can have more than one point light.

21.93.2 Options within the command

Name

Specifies a name for the light so that it can be identified by the LIGHTLIST command.

Intensity factor

Specifies the intensity of the light.

Status

Toggles the use of this light.

Photometry

Specifies the photometric properties of the light.

Intensity

Sets the intensity expressed in candela.

Flux

Sets the luminous flux expressed in lumens.

Illuminance
Sets the illuminance expressed in lux or foot-candles.

Distance
Sets a distance expressed in drawing units.

Color
Sets the color.

?
Lists the names of available colors.

Kelvin
Sets a color temperature expressed in degrees Kelvin.

shadoW
Specifies the look of shadows cast by this light.

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges. Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources.

Shape
Sets the shape of the light.

Disk
Sets the radius of the disk.

Rect
Sets the length and width of the rectangle.

sAmples
Specifies the sampling size of the shadow.

Note: Larger numbers are more accurate but take longer to render.

Visible
Toggles the shape visibility.

Attenuation
Specifies how the illumination falls off with distance from the light.

attenuation Type
Specifies the type of attenuation.

None
No attenuation so that the distance to the light source has no influence.

Inverse linear
Attenuation is the inverse of the linear distance from the light.

Note: At a distance of 2 units from the light source, light is half as strong. At a distance of 4 units, light is one quarter as strong.

inverse Squared
Attenuation is the inverse of the square of the distance from the light.

Note: At a distance of 2 units, light is one quarter as strong. At a distance of 4 units, light is one sixteenth as strong.

Use limits
Toggles whether the extent of the illumination is limited.

attenuation start Limit
Defines the point where the light starts to shine, measured from the center of the light.

attenuation End limit
Defines the point where the light stops to shine, measured from the center of the light.

filterColor
Specifies the color of the light.

Index color
Specifies an Index color.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness.

Colorbook
Specifies a bookcolor name.

21.94 POLYGON command

Creates a closed polyline in the shape of a polygon.

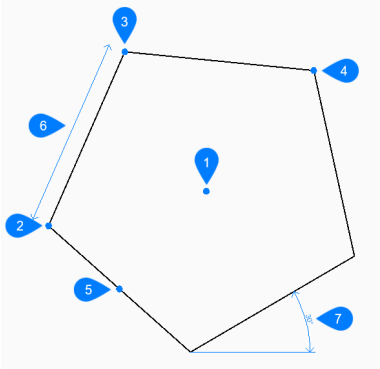
Shape Lite Pro Mechanical BIM

Icon: 

Alias: POL

21.94.1 Description

Creates a closed polyline in the shape of an equal-sided polygon. Options allow you to specify the center, number of sides, edge length and angle, and the distance from the center to the vertex or midpoint of an edge.



1 Center of polygon

2 First endpoint edge

3 Second endpoint edge

4 Vertex

5 Midpoint of side

6 Edge length

7 Angle of polygon

21.94.2 Method

This command creates a polygon with these basic steps:

- Set number of sides.
- Set center of polygon.
- Select vertex point.

21.94.3 Options within the command

Set number of sides
Allows you to begin creating a polygon by specifying the number of sides between 3 and 1024.

Set center of polygon
Specifies the center point of the polygon.

Select midpoint of side
Specifies the location of the midpoint of a polygon line segment. The location of the midpoint defines the size and angle of the polygon.

Width of line
Specifies the width of the polygon line segments. All segments have the same width.

Multiple Polygons
Creates multiple polygons with the same size and orientation when using the **Set center of polygon** option. Continue placing polygons until you press Enter to end the command.

specify by Edge
Specifies the endpoints of one edge of the polygon to define its size and angle.

specify by Vertex
Specifies the location of a vertex of the polygon. The location of the vertex defines the size and angle of the polygon.

21.95 POLYSOLID command

Creates a 3D solid in the shape of a wide, extruded polyline.

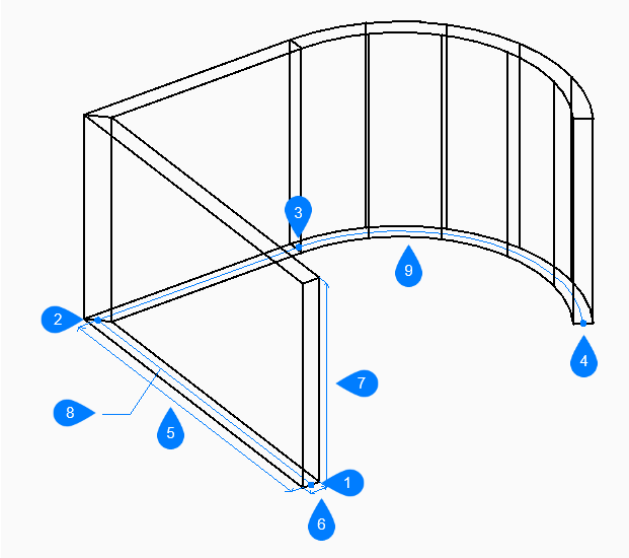
Shape Lite Pro Mechanical BIM

Icon:

Alias: PSO

21.95.1 Description

Creates a 3D solid in the shape of a wide, extruded polyline with multiple line and arc segments. Options allow you to specify the width, height and justification.



1 Start
2 Next
3 Next
4 End
5 Length
6 Width
7 Height
8 Base path
9 Arc segment

21.95.2 Method

This command has 2 methods to begin creating a polysolid:

- Start point
- Entity

Create a polysolid by specifying the start point and next vertex of the base path, and the height of the polysolid.

Note: You can continue adding unlimited vertices until you press **Enter** key to end the command.

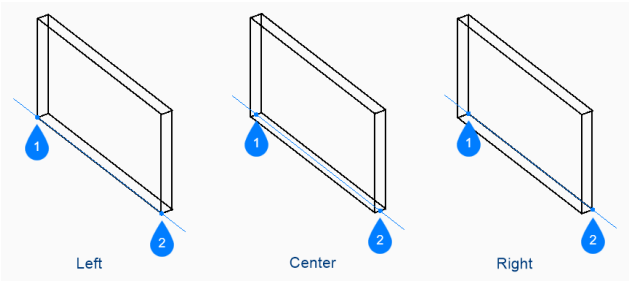
21.95.3 Options within the command

Height
Specifies the height of the polysolid.
The height is saved in the PSOLHEIGHT system variable.

Width
Specifies the width of the polysolid.
The width is saved in the PSOLWIDTH system variable.

Justification
Specifies the position of the polysolid with respect to the base path.

- **Left**: the left edge of the polysolid follows the base path.
- **Right**: the right edge of the polysolid follows the base path.
- **Center**: the center of the polysolid follows the base path.



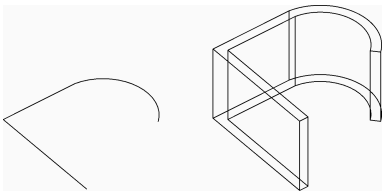
1 Start

2 End

Note: When the Hot Key Assistant (HKA) is enabled, the following widget allows you to dynamically switch between left, center, and right justification by pressing the **Ctrl** key.



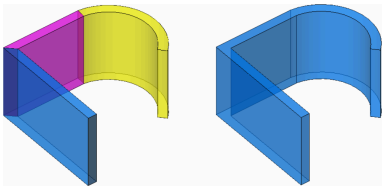
Entity
Allows you to select a 2D entity (line, open or closed polyline, arc, circle, ellipse, elliptical arc, or splines) as base path of the polysolid.



Separate solids

Specifies whether polysolid creates an individual 3D solid for each segment or a single 3D solid for all the segments.

- **On**: create individual solids.
- **Off**: create a single solid.



Dynamic

Specifies whether you are prompted to specify the height when creating a polysolid.

- **On**: you are prompted to specify the height.
- **Off**: the height is defined by the PSOLHEIGHT system variable.

Draw lines

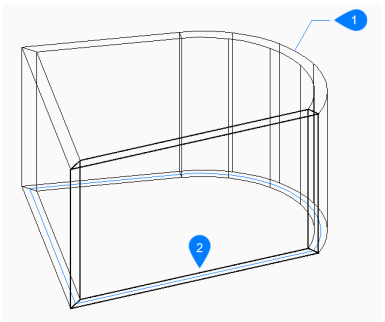
Draws line segments to the polysolid profile by specifying the next point.

The line segment is drawn tangent to the previous segment.

Close

Automatically draw a polysolid line segment from the end point of the last segment to the start point of the first segment.

At least three points have to be specified to use this option.



1 Last drawn segment

2 Close

Distance

Specifies the length and angle of a polysolid segment.

Follow

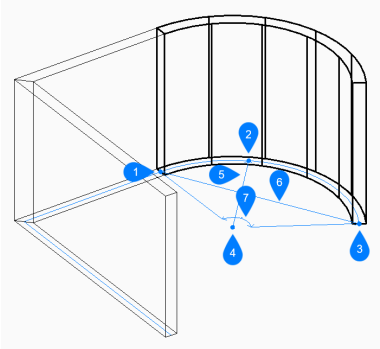
Creates a line segment from the last drawn arc or line segment, following its angle.

Draw arcs
Draws arc segments to the polysolid profile by specifying the end of arc.
The arc segment is drawn tangent to the previous segment.

CLose
Automatically draws a polysolid arc segment from the last point specified to the starting point of the polysolid.
At least three points have to be specified to use this option.

Direction
Specifies the direction of the arc segment by setting a starting tangent direction and an endpoint.

Second point
Specifies a point along the polysolid arc's circumference.



1 First point

2 Second point

3 End of arc

4 Center

5 Radius

6 Length of chord

7 Angle

Undo
Undoes the last polysolid segment and continue drawing from its previous start point.

21.96 PREVIEW command

Displays the Preview window so that you can check how the drawing looks before plotting it.

Shape Lite Pro Mechanical BIM

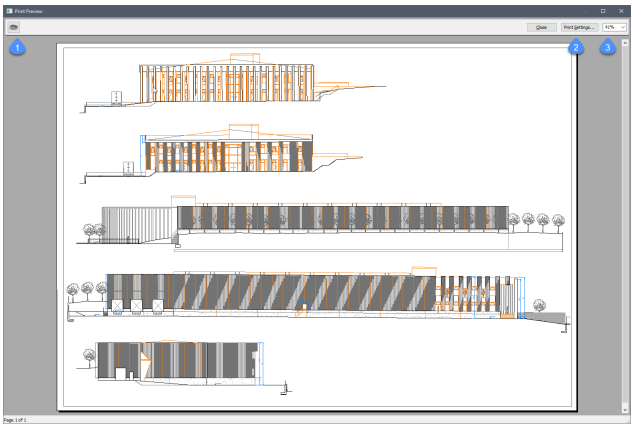
Icon:

Alias: PREVIEW, PRE

Note: This command does not work with the "None" printer; use the PRINT or PAGESETUP command to first specify a printer.

21.96.1 Description

Displays a window:



Use the mouse scroll wheel to zoom and use the scroll bars to pan.

1. Print
Prints the drawing; does not display the PRINT dialog box. See PRINT command.

2. Print Settings
Displays the PRINT dialog box. See PAGESETUP command.

3. Zoom
Changes the size of the preview.

21.97 PRINT command

Prints the drawing to printers, plotters or files.

Shape Lite Pro Mechanical BIM

Icon:

21.97.1 Description

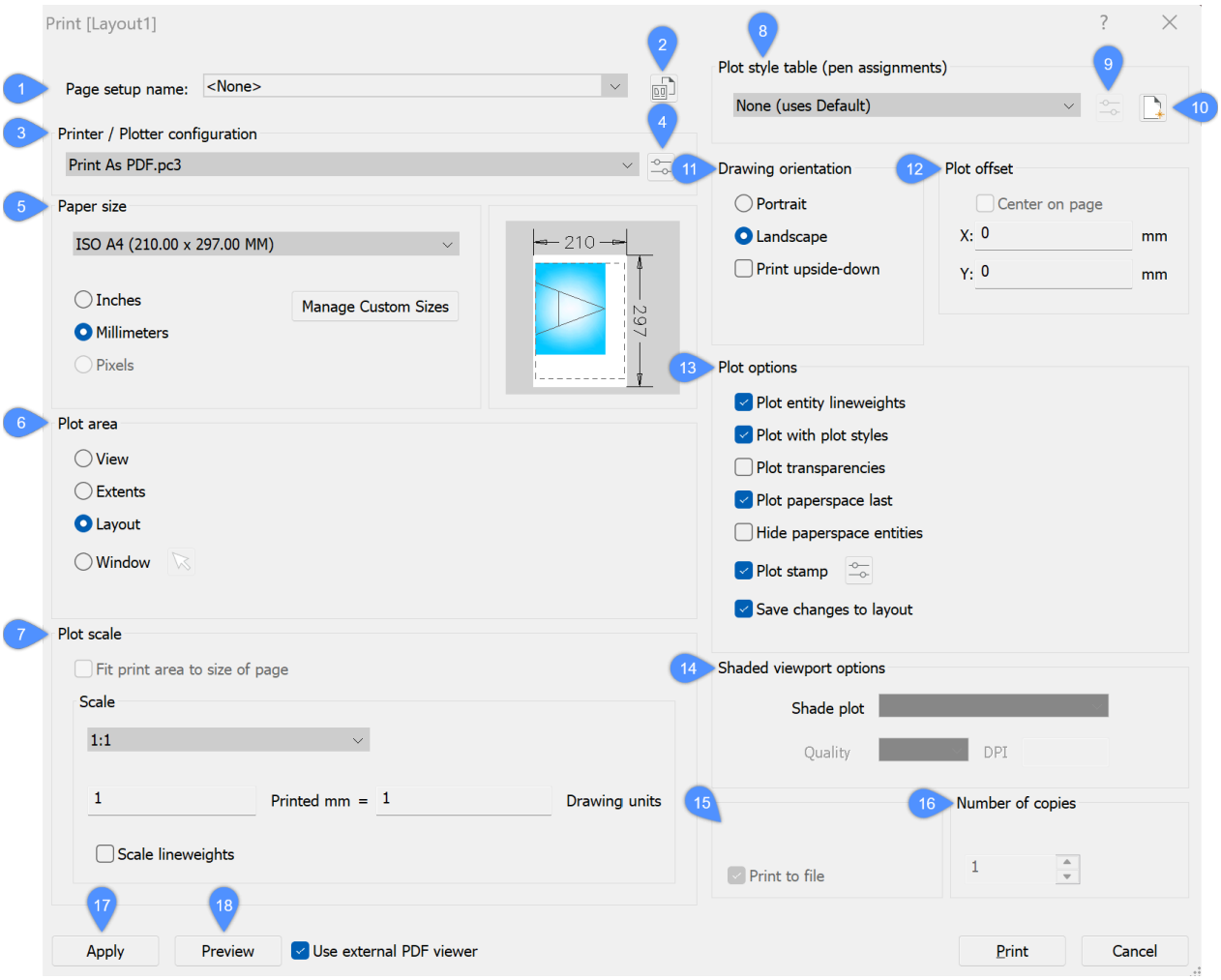
Opens the **Print** dialog box to specify print options and to preview or print the current drawing.

Note: This command adds a watermark to print output obtained using an Academic license.

The **Print** dialog box allows you to print and preview drawings to plotters and files.

Remember:

- On macOS and Linux it is not possible to print to system printers. It is only possible to print to the **Print As PDF.pc3** printer. So, a PDF document will be generated, which needs to be sent to the printer for physical printing.
- On macOS and Linux there is a relevant PAGESETUP command.



1 Page setup name
2 Create page setup
3 Printer/Plotter configuration
4 Edit plotter configuration
5 Paper size
6 Plot area
7 Plot scale
8 Plot style table
9 Edit Plot Style
10 Create New Plot Style
11 Drawing orientation
12 Plot offset
13 Plot options
14 Shaded viewport options

15 Print to file

16 Number of copies

17 Apply

18 Preview

21.97.2 Page setup name

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.

21.97.3 Create page setup

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

21.97.4 Printer/Plotter configuration

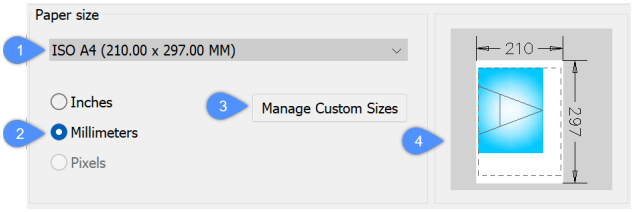
Allows you to select from the drop-down list the printer or plotter.

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.

21.97.5 Edit plotter configuration

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3 files. See the PLOTTERMANAGER command.

21.97.6 Paper size



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.

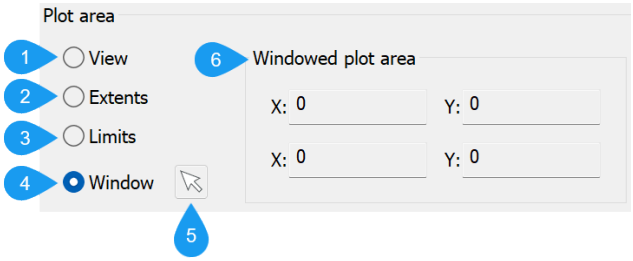
Manage Custom Sizes

Note: This option is available for the PDF, PNG, TIF, BMP, and JPG outputs.

Manages custom paper sizes for the PC3 printer selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.
You can attach a predefined plotter model parameter (PMP) file to a plotter configuration (PC3) file.

Preview
This section indicates the size, position and orientation of the current plot area on the selected paper size.

21.97.7 Plot area



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.

21.97.8 Plot scale

Scales the drawing to fit it to the paper:

- **Fit print area to size of page**: 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.

21.97.9 Plot style table

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.

21.97.10 Edit Plot Style

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

21.97.11 Create New Plot Style

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).

21.97.12 Drawing orientation

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.

21.97.13 Plot offset

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.

21.97.14 Plot options

- **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.

21.97.15 Shaded viewport options

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.

21.97.16 Print to file

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

21.97.17 Number of copies

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

21.97.18 Apply

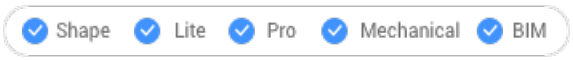
Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

21.97.19 Preview

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.

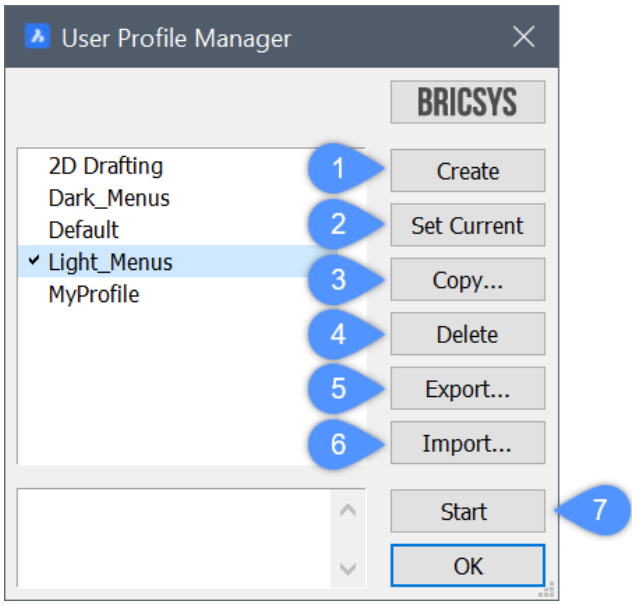
21.98 PROFILEMANAGER command

Opens the **User profile manager** dialog box.



21.98.1 Description

Opens the **User profile manager** dialog box to create, copy, delete, import and export user profiles.



1 Create

2 Set Current

3 Copy...

4 Delete

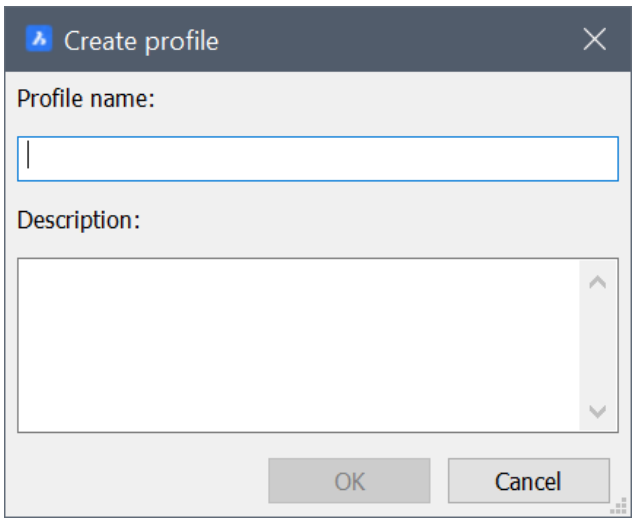
5 Export...

6 Import...

7 Start

21.98.2 Create

Allows you to create a new profile, using built-in default settings. The **Create profile** dialog box displays:



Profile name

Allows you to type a name for the new profile.

Description
(Optional) Allows you to type a description for the new profile.

21.98.3 Set Current

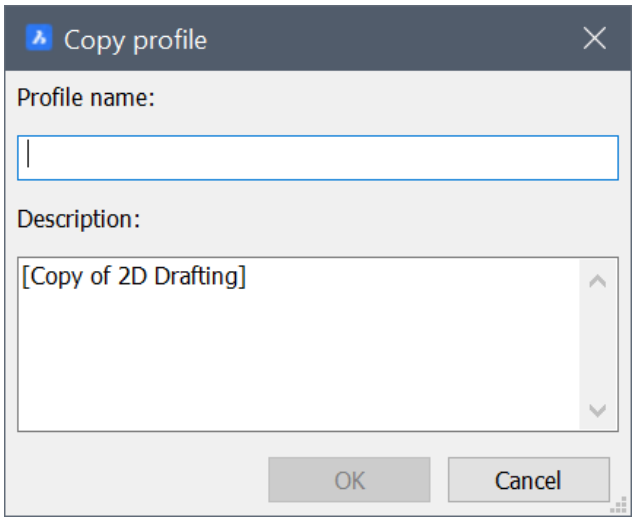
Sets the selected profile as the current profile.

Note:

- You can also double click a profile to set it current.
- The current profile is marked in the list with a check mark.

21.98.4 Copy

Creates a new profile as a copy of an existing profile. The **Copy profile** dialog box displays:



21.98.5 Delete

Deletes the selected profile.

Note: Deleting the current profile is not possible. When trying to do so, a dialog box displays that allows to reset all settings to factory defaults.

21.98.6 Export...

Exports the selected profile to an .arg (profile file) or .reg (registry file) to a specified folder.

Note: Exporting and importing a profile allows you to migrate user preferences from one computer to another.

21.98.7 Import...

Imports saved profiles.

21.98.8 Start

Starts a BricsCAD session using the selected profile.

Note:

- This option is only available when the command is launched when executing **profilemanager_app.exe** in the BricsCAD installation folder. By default, the path is: *C:\Program Files\Bricsys\BricsCAD V25 en_US*.
- If the SINGLETONMODE preference variable is set to On, it is not possible to have multiple BricsCAD sessions open simultaneously.

21.99 PROJECTGEOMETRY command

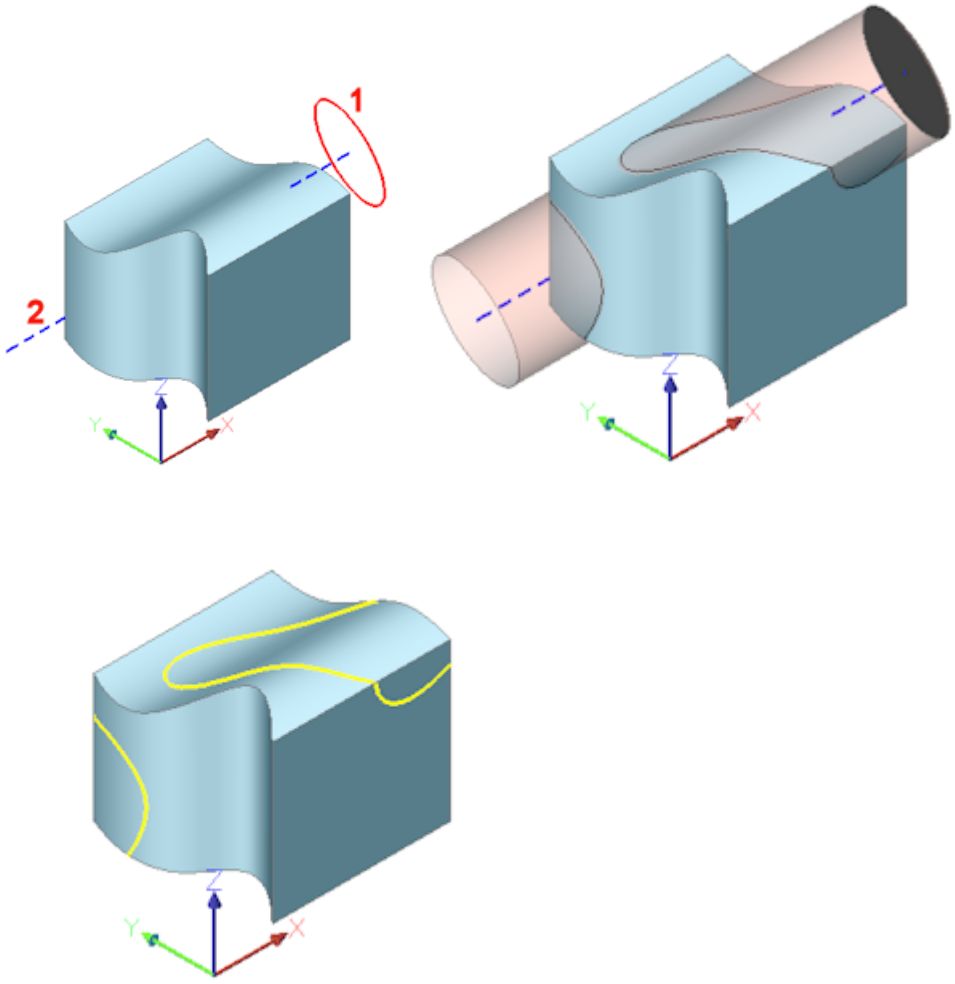
Projects 2D geometry.

Shape Lite Pro Mechanical BIM

Icon:

21.99.1 Description

Projects 2D geometry onto regions, surfaces, or 3D solids to create additional edges (line work).



(1) 2D entity; (2) 3D solid; blue line = projection direction; pink tube = projection; yellow curves = new edges

21.99.2 Options within the command

set PROjection direction
Defines the projection direction. By default, 2D geometry is projected orthogonally onto the receiving entity.

View
Sets the projection direction perpendicular to the current view orientation.

UCS
Uses the Z-axis of the current UCS.

Points
Specify the start and end point of the projection direction.

21.100 PROPERTIES command

Opens the **Properties** panel in Properties mode.

Shape Lite Pro Mechanical BIM

Icon:

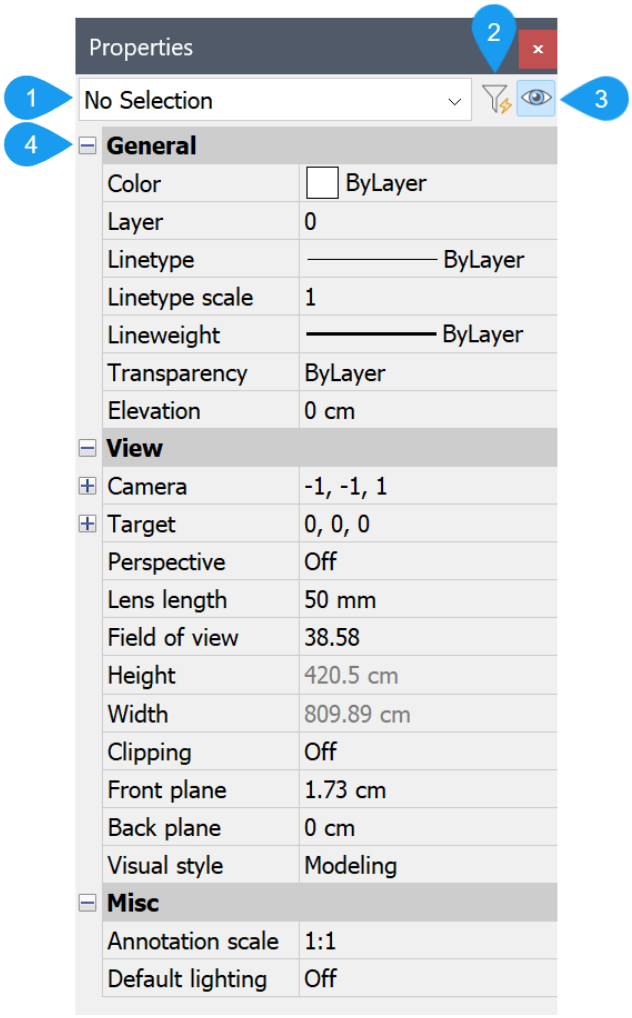
Alias: CH, DDCHPROP, DDMODIFY, MO, PR, PROPS

21.100.1 Description

Opens the **Properties** panel to display it in the current workspace. The **Properties** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Properties** panel can be either floating, docked or stacked.

The **Properties** 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 Properties list

Entities list

Displays the type and number of selected entities.

Note: The PROPOBJLIMIT system variable defines the limit of entities displayed in the **Properties** panel to improve performance. If the number of selected entities exceeds the limit, a warning message appears, and you are allowed to choose one of the options: **Ignore limit** or **Change limit**.

Properties/Quick Select toggle

Switches between **Properties** and **Quick Select** mode.

Property Preview toggle

Turns the PROPERTYPREVIEW setting On or Off.

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.

21.100.2 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.

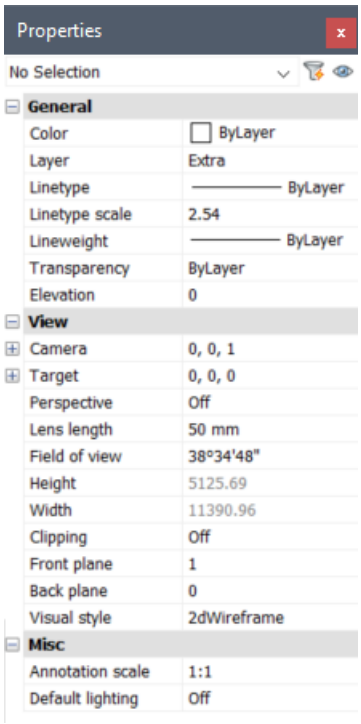
Entities list

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

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 **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
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
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.

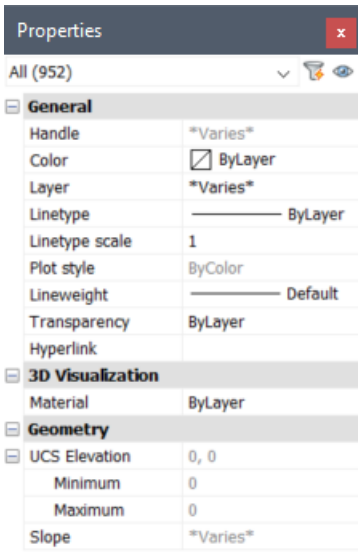
Default lighting
Describes the default lighting status (DEFAULTLIGHTING system variable). You can turn **Default lighting** On or Off from the drop-down list.

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*.

Note:

- When area calculations take longer than three seconds, temporary yellowish node labeled "Computing..." is inserted in **Properties** panel. Even if you interrupt the calculation by pressing ESC in the **Properties** panel, the entities remain highlighted and their properties are available for the user.
- The mass properties of 3D solids and surfaces are visible in the **Properties** panel when multiple entities are selected.

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. Besides the properties that are available when no entity is selected, there are:

Handle (read only)

Describes the unique hexadecimal value by which the selected entity is identified in the DWG database.

Plot style

Describes the plot style of the selected entity. You can specify a new plot style from the drop-down list. Note that this property is only available when plot styles are enabled. Also see the PLOTSTYLE command

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

Describes 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

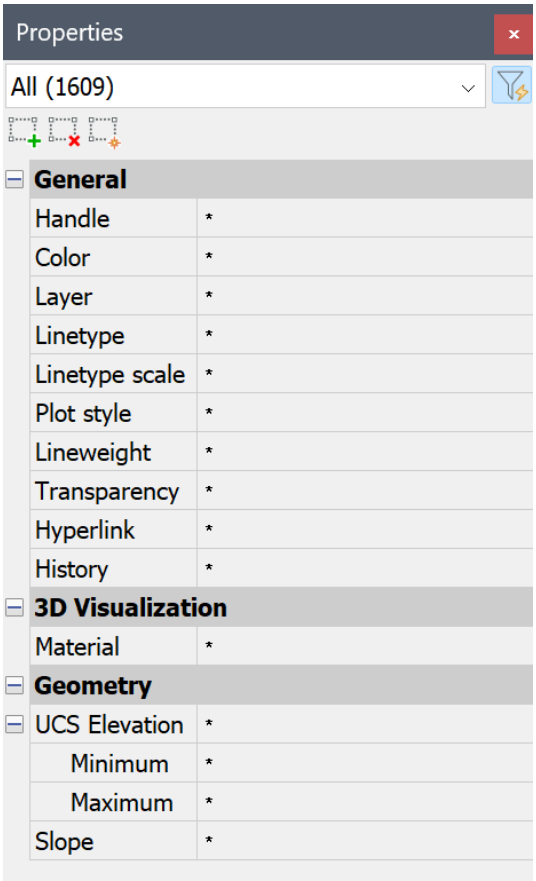
Describes the geometric characteristics of the selected entities. These properties can vary for different types of entities.

Note: When you click a numeric field, a button that opens the **Calculator** panel will appear on the right side.

Click the Apply button from the Calculator panel to paste the calculated value in the numeric field from the Properties panel.

21.100.3 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 operators.



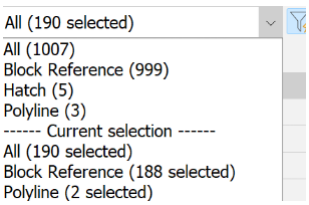
Selection tools

Creates a selection set based on entity properties.

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.

Note: Click the drop-down button in the Entities list to see the current selection.



Remove from current selection set

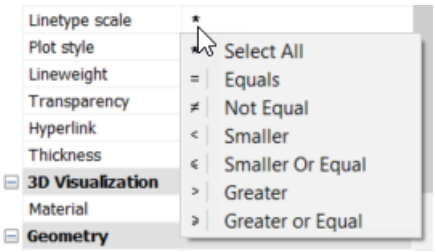
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists.

Clear current selection and create to new selection

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 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

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.

21.101 PROPERTIESCLOSE command

Closes the **Properties** panel.

Shape Lite Pro Mechanical BIM

Alias: PRC

21.101.1 Description

Closes the **Properties** panel to hide it from the current workspace. If the **Properties** panel is stacked when you close it, the **Properties** tab or icon is removed from the stack.

21.102 PROPULATE command (Express Tools)

Lists, removes or updates drawing properties data.

Shape Lite Pro Mechanical BIM

Icon: 

21.102.1 Description

Allows you to list, remove or update drawing properties data. Using a template, the drawing properties data can be added to folders of drawings. If a customized template is used it is possible to extract attribute values from title blocks to the drawing properties. The information that can be extracted are attribute values of blocks and lists of attached xrefs, images and fonts.

21.102.2 Options within the command

Active template
Opens the **Select Propulate Template File** dialog box that allows you to select a propulate template file that is used to update drawing properties fields.

Edit template
Opens the **Edit Propulate Template** dialog box that allows you to create and edit propulate template files.

List
Allows you to choose between **Current drawing** and **Other drawings**.

Current drawing
Displays the drawing properties for the current drawing.

Other drawings
Asks you to specify a search directory and a drawing name.

> ***Note:*** If you add a * in the directory, all the drawings starting with the directory part before the * will be searched and optionally its subfolders. For example, if you use *C:\Drawings\Plan**, all the drawings with a name that start with Plan will be searched in the *C:\Drawings folder*.

Search subdirectories?
Choose **Yes** to search in subfolders of the directory.

Remove
Removes the drawing properties from the current drawing or a specified drawing.

Update
Propulates the drawing properties from the current drawing or a specified drawing using the active template.

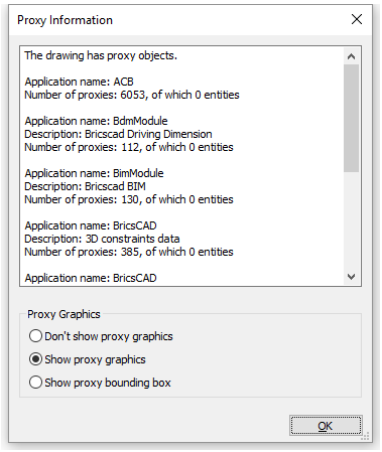
21.103 PROXYINFO command

Opens the **Proxy information** dialog box.

Shape Lite Pro Mechanical BIM

21.103.1 Description

Opens the **Proxy information** dialog box to view information about proxy entities and toggle their display in the current drawing.



21.103.2 Proxy Graphics

Toggles the display of proxy graphics in the drawing:

- **Don't show proxy graphics**: turns off the display of proxy graphics.
- **Show proxy graphics**: turns on the display of proxy graphics
- **Show proxy bounding box**: displays a rectangle or cube instead of the proxy.

21.104 PSBSCALE command (Express Tools)

Sets or updates the scale of block references relative to paper space.

Shape Lite Pro Mechanical BIM

21.104.1 Method

For the inserted blocks in model space, you can specify the size they should be displayed in paper space.

Note: Use the **Update** option to adjust previously set entities, in case the zoom factor changes.

21.104.2 Options within the command

Set

Specifies the X and Y scale factor relative to paper space.

XYZ

Specifies the X, Y and Z scale factor relative to paper space.

Update

Updates the scale of the selected block reference.

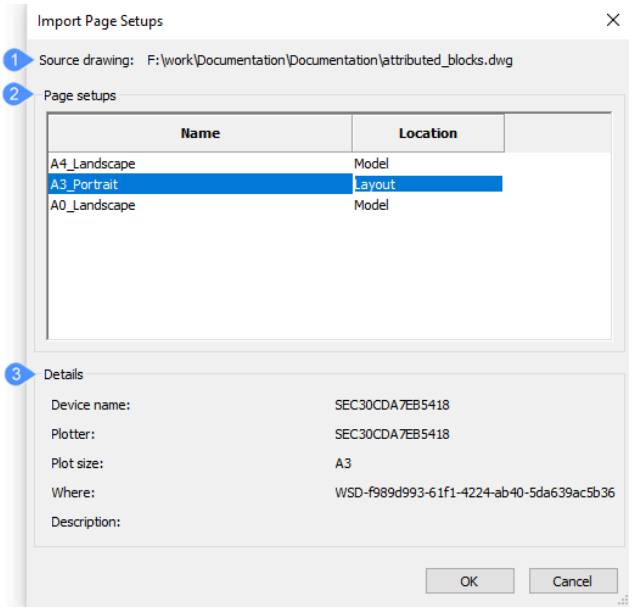
21.105 PSETUPIN command

Opens the **Import Page Setups** dialog box.

Shape Lite Pro Mechanical BIM

21.105.1 Description

Opens the **Select page setup from file** dialog box to select a DWG, DWT, or DXF file from which to import page setups. After selecting the file and choosing **Open**, the **Import Page Setups** dialog box displays. It enables you to choose which page setups from the selected file you wish to import into the current drawing.



1 Source Drawing

2 Page setups list

3 Details

21.105.2 Source Drawing

Displays the source of the drawing used to import the page setup(s).

21.105.3 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.

21.105.4 Details

Includes information about the device name, plotter, plot size, location and a description field.

21.106 -PSETUPIN command

Imports page setup definitions.

Shape Lite Pro Mechanical BIM

21.106.1 Description

Imports page setup definitions from another drawing through the Command line.

Note: Page setups define how a drawing is printed and are used by the PRINT and PUBLISH commands.

21.106.2 Method

Run the command to open the **Select Page Setup From File** dialog box.

21.107 PSPACE command

Switches from model space and paper space viewports.

Shape Lite Pro Mechanical BIM

Alias: PS

21.107.1 Description

Switches from model space to paper space viewport in a layout tab. This command has a shortcut menu that opens by right-clicking on **P:Layout** on the Status Bar.

Note: This command operates only when a layout tab is in paper space mode.

21.108 PSTSCALE command (Express Tools)

Sets or updates the scale of text entities relative to paper space.

Shape Lite Pro Mechanical BIM

21.108.1 Description

Sets or updates the paper space height of single-line and multiline text entities from model space in a layout viewport.

21.108.2 Options within the command

Set

Allows you to specify the text height in paper space units.

Update

Updates the scale of the selected block reference.

21.109 PUBLISH command

Prints the content of a DSD file.

Shape Lite Pro Mechanical BIM

Icon: 

21.109.1 Description

Opens the **Publish** dialog box to batch print a specified set of drawings.

Note: This command adds a watermark to publish output obtained using an Academic license.

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.

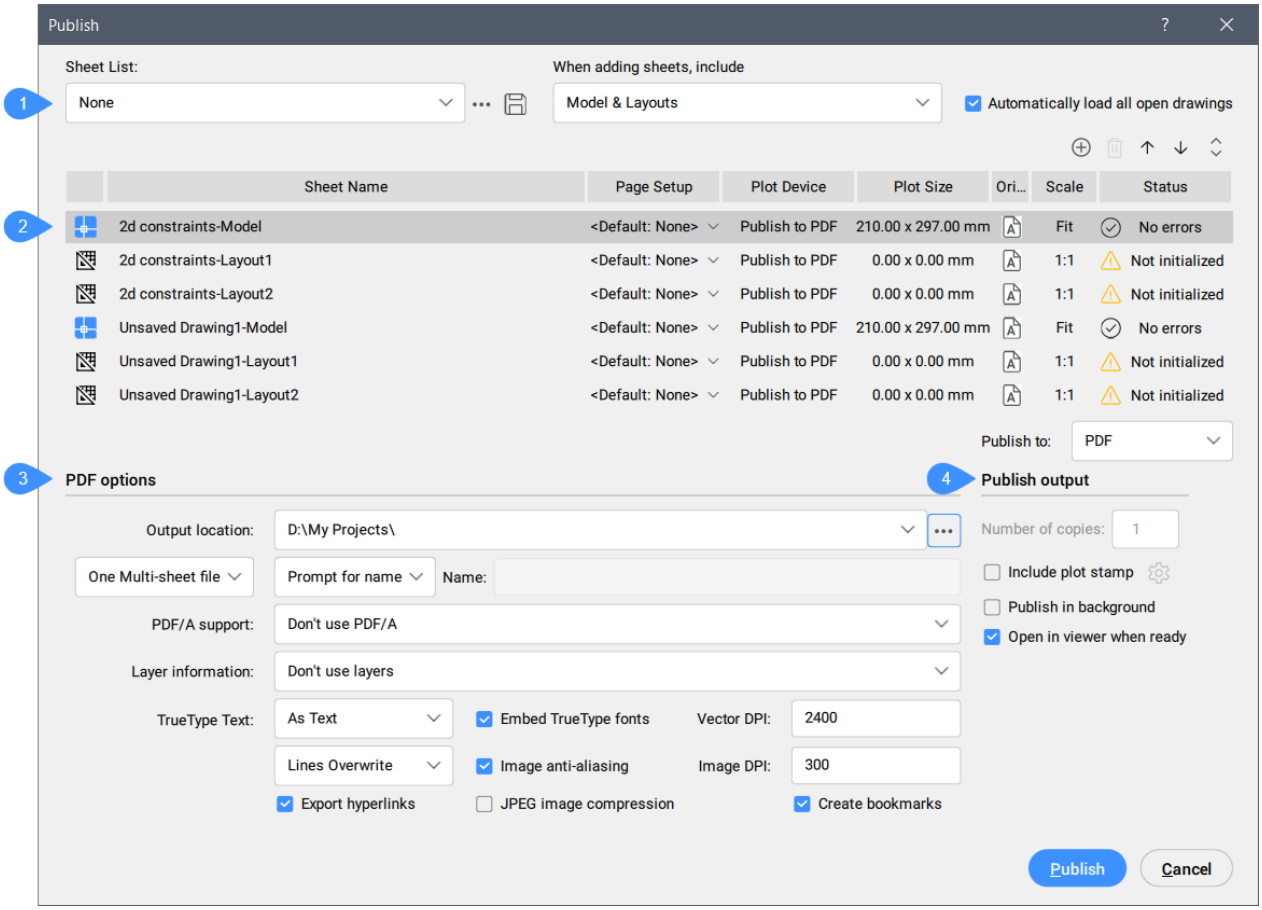
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command:

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"

Note: In the **Page Setups** section of the **Drawing Explorer** dialog box, set the printer you use in the **Device name** field.



1 Sheets selection

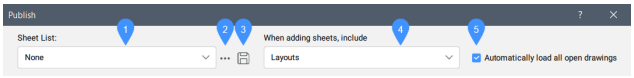
2 Sheet list preview

3 PDF options

4 Publish output

21.109.2 Sheets selection

Displays a list of sheets. If you have a *.dsd file (drawing sheets for publishing), select it and the **Load Sheet List** dialog box appears.



Sheet List

Specifies the selected sheet list. A sheet list can be selected in the drop-down menu.

Load Sheet List

Loads a saved sheet list. Opens the **Load Sheet List** dialog box. If the current list is not saved, you are prompted to save it.

If you are loading 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.

Save Sheet List

Saves the current sheet list to a *.dsd (Drawing Set Description) file. The **Save Sheet List** dialog box opens.

When adding sheets, include

Determines which sheets are included when loading new sheets using the **Add Sheets** button. Click the drop-down button to choose between **Model**, **Layouts**, and **Model & Layouts**.

- **Model**: includes only model tabs of the selected drawings.
- **Layouts**: includes only layout tabs of the selected drawings.
- **Model & Layouts**: includes both model and layout tabs of the selected drawings.

Automatically load all open drawings

Determines how open drawings are handled:

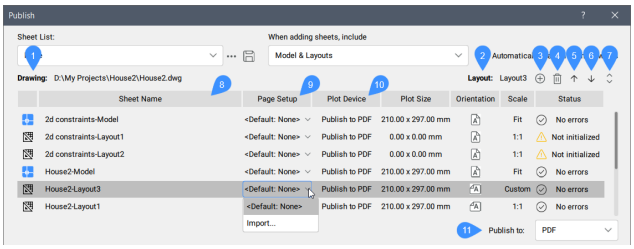
- **On** (checked): adds all model and/or paper space layouts to the list of sheets to be plotted.
- **Off** (unchecked): adds the model and/or paper space layouts of the current drawing only.

This option also sets the value of the PUBLISHALLSHEETS system variable.

Note: The PUBLISHCOLLATE system variable determines whether the plotting process of a sheet set can be interrupted by other plot jobs (for plot driver that support the multi-sheet plotting or printing option).

21.109.3 Sheet list preview

Displays characteristics of all selected sheets.



Drawing

Specifies the path of the drawing file selected in the list.

Layout
Specifies the layout of the drawing selected in the list.

Add Sheets
Displays the **Select Drawing(s)** dialog box to load sheets.

Remove Sheets
Removes the selected sheet from the sheet list.

Move Sheet(s) up
Moves the selected sheet one position up in the list.

Move Sheet(s) down
Moves the selected sheet one position down in the list.

Reverse Sheet Order
Reverses the order of the sheets list.

Page Setup
Specifies the Page Setup name of each sheet. Click the **Page Setup** field, then click the down arrow to select a different page setup or choose **Import...** to import page setups of another drawing. The **Import Pagesetups** dialog box opens.

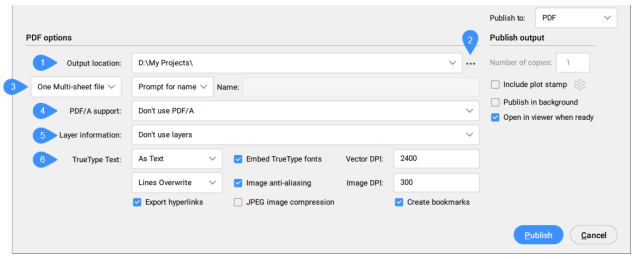
Plot Device
Displays the name of the plot device, as defined in the **Page Setup** dialog box of the sheet.

Publish to:

PDF
The **PDF options** section is displayed.

Plotter named in page setup
Publishes to the plotter displayed in the **Plot Device** field (set via the **Page Setup** dialog box).

21.109.4 PDF options



Output location
Displays the path where the PDF will be saved.

Browse button
Opens the **Choose a folder** dialog box to change the path.

Output file
Select an option:

Single–sheet files
Each layout is published to a separate PDF file. The file names are: <drawing>_<layout>.pdf.

Name options
Select an option:

- **Use sheet name**: the PDF will have the same name as the sheet file.
- **Use layout name**: the PDF will have the same name as the layout.
- **Use drawing name**: the PDF will have the same name as the DWG file.

One Multi–sheet file
Layouts are published to a single PDF file. You can specify a name or be prompted for a name when the publishing procedure starts.

Name options
Select an option:

- **Prompt for 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 **Name** field becomes available. Insert the PDF name.
- **Generate name**: the PDF will have the same name as the drawing.

PDF/A Support
Allows you to select an option from the drop-down list: **Don't use PDF/A** (default option), **Use version PDF/A-1b**, **Use version PDF/A-2b**, **Use version PDF/A-3b**.

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.

Merge control
Determines 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.

Export hyperlinks
If checked, it allows you to export hyperlinks in the PDF.

Embed TrueType fonts
If the font is not embedded, a substitute font might be used by the PDF viewer.

Note: Licensed fonts cannot be embedded.

Image anti-aliasing
If the resolution of an image is too low to match the Image DPI setting, 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.

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.

Vector DPI
Controls the resolution of vector graphics and gradients.

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.

Create bookmarks
In case of choosing **One Multi-sheet file** option, it creates bookmarks for each sheet in the output file.

21.109.5 Publish output

Specifies to what plotter the sheets should be published to.

Note: 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.

Edit 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 is opened in the viewer when it is ready.

Note: This option is not available when **Publish to** is set to **Plotter named in page setup**, only when **Publish to** is set to **PDF**.

21.110 -PUBLISH command

Prints the content of a DSD file.

Shape Lite Pro Mechanical BIM

21.110.1 Description

Prints the content of a Drawing Set Description (*.dsd) file at the Command line.

Note: Create the DSD file with the PUBLISH command. The DSD file specifies the file name, layouts, page setups, plotter and printer names, orientation, plot scale, number of copies, optional plot stamp, and the order in which to plot the files.

Note: This command adds a watermark to publish output obtained using an Academic license.

21.110.2 Method

Select a DSD file in the **Select Sheet List** dialog box.

Note: If the BACKGROUNDPLOT system variable is 2 or 3, the sheets are printed in the background.

21.111 PURGE command

Removes unused named entities from drawings.

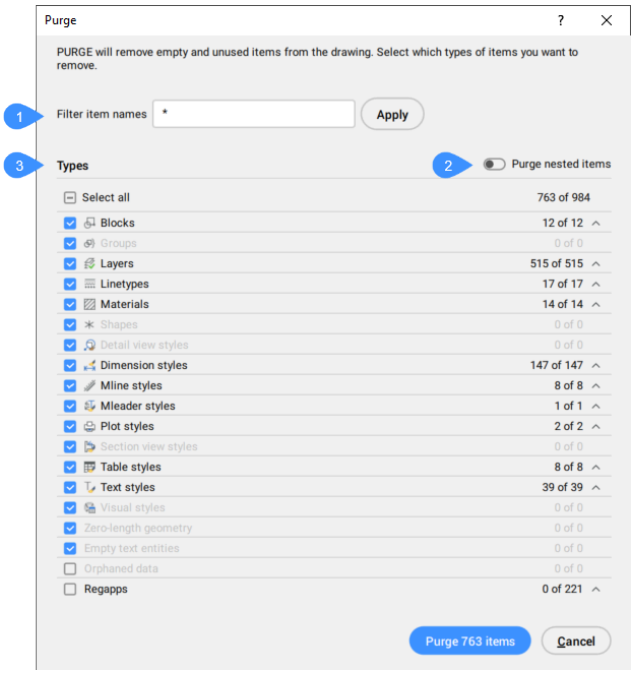
Shape Lite Pro Mechanical BIM

Icon:

Alias: PU

21.111.1 Description

Opens the **Purge** dialog box to select unused named entities to remove from the drawing.



1 Filter item names

2 Purge nested items

3 Types

21.111.2 Filter item names

Apply a filter to the item names in the drawing.

21.111.3 Purge nested items

When enabled, it purges all nested items. Therefore, it is not needed anymore to run PURGE multiple times.

21.111.4 Types

You can toggle on/off the different types of items that will be removed.

Blocks
Removes unused block definitions from the current drawing.

Detail view styles
Removes unused detail view styles.

Dimension styles
Removes unused dimension styles from the current drawing.

Groups
Removes unused named groups from the current drawing.

Layers
Removes unused layers from the current drawing.

Linetypes
Removes unused linetypes from the current drawing.

Materials
Removes unused materials from the current drawing.

Mline styles
Removes unused multiline styles from the current drawing.

Multileader styles
Removes unused multileader styles from the current drawing.

Plot styles
Removes unused plot styles from the current drawing.

Regapps
Removes unused regapp keys from the current drawing.

> ***Note:*** Regapp (registered application) keys are used to attach additional information to entities in drawings by third-party applications.

Section view styles
Removes unused section view styles from the current drawing.

Shapes
Removes unused shapes loaded into the drawing with the LOAD command.

Table styles
Removes unused table styles from the current drawing.

Text styles
Removes unused text styles from the current drawing.

Visual styles
Removes unused user-defined visual styles from the current drawing.

Note: Only user-defined visual styles can be purged. Hard-coded visual styles cannot be purged.

Zero-length geometry
Removes geometry without length.

Empty text entities
Removes empty text entities from the current drawing.

Orphaned data
Performs a drawing scan and removes unused DGN linestyles data.

21.112 -PURGE command

Removes unused named entities from drawings.

Shape Lite Pro Mechanical BIM

Alias: -PU

21.112.1 Method

Specify which type of entities to purge.

21.112.2 Options within the command

BAtch all
Purges all unused named entities and nested entities from the drawing without prompts.

purge All
Purges all unused named entities from the drawing following the prompts.

Yes
Purges each unused named entity when you confirm.

No
Purges all unused named entities and nested entities without confirmation.

Blocks
Removes unused block definitions from the current drawing.

DEtail view styles
Removes unused detail view styles.

Dimension styles
Removes unused dimension styles from the current drawing.

Groups
Removes unused named groups from the current drawing.

LAyers
Removes unused layers from the current drawing.

LineTypes
Removes unused linetypes from the current drawing.

MAterials
Removes unused materials from the current drawing.

MLine styles
Removes unused multiline styles from the current drawing.

MUltileader styles
Removes unused multileader styles from the current drawing.

Plot styles
Removes unused plot styles from the current drawing.

Regapps
Removes unused regapp keys from the current drawing.

Note: Regapp (registered application) keys are used to attach additional information to entities in drawings by third-party applications.

SEction view styles
Removes unused section view styles from the current drawing.

SHapes
Removes unused shapes loaded into the drawing with the LOAD command.

Table styles
Removes unused table styles from the current drawing.

text STyles
Removes unused text styles from the current drawing.

Visual styles
Removes unused user defined visual styles from the current drawing.

Note: Only user defined visual styles can be purged. Hard coded visual styles cannot be purged.

Zero-length geometry
Removes geometry with no length.

Empty text entities
Removes empty text entities from the current drawing.

Orphaned data
Performs a drawing scan and removes unused DGN linestyles data.

21.113 PYRAMID command

Creates a 3D Solid in the shape of a pyramid.

Shape Lite Pro Mechanical BIM

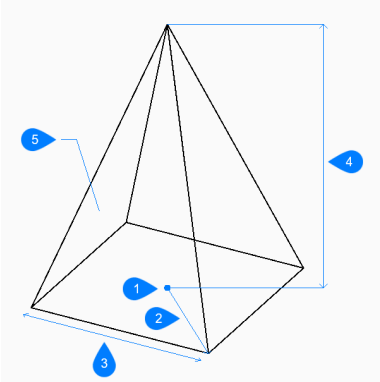
Icon:

Alias: PYR

Note: In BricsCAD Lite, which doesn't support 3D Solids, the PYRAMID command launches the AI_PYRAMID command.

21.113.1 Description

Creates a 3D Solid in the shape of a pyramid with at least three sides. Choose from a combination of options including center, edge, radius, sides, height, and axis. The pyramid can have a pointed or flat top.



1 Center

2 Radius

3 Edge

4 Height

5 Side

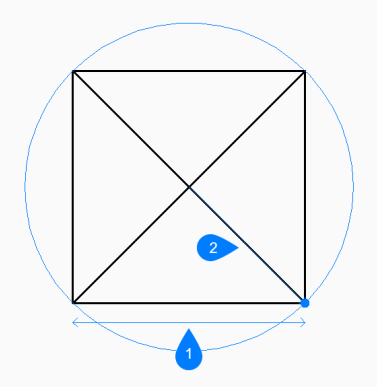
21.113.2 Method

This command has 2 methods to begin creating a pyramid:

- Specify center point
- Edge

Create a pyramid by specifying the center point, base radius and the height.

Note: Base radius represents the distance from the center to a vertex as if the base is inscribed in a circle.



1 Center

2 Radius

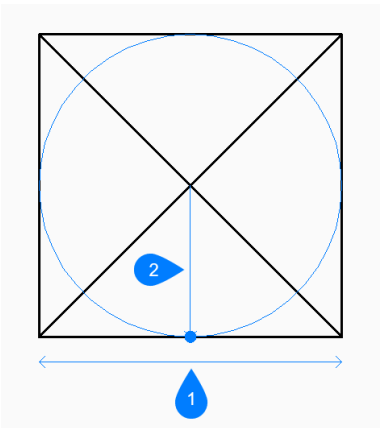
21.113.3 Options within the command

Edge
Allows you to begin creating a pyramid by specifying the first endpoint for an edge of the base.

Specify second point of edge
Specifies the second point of the edge to define its length and angle in the xy-plane.

Sides
Specifies the number of sides or vertices for the base.

Circumscribed
Specifies the distance from the center to a midpoint on an edge as if the base is circumscribed about a circle.



1 Edge

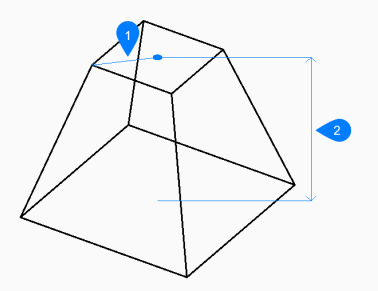
2 Radius

2Point
Specifies the height of the pyramid as the distance between any two points.

Axis endpoint
Specifies the axis endpoint to define the height and orientation of the pyramid in 3D space. The center of the base is used as the other axis endpoint.

Top radius

Specifies the distance from the center of the top face to a vertex as if the top face is inscribed in a circle. Any radius larger than zero (0) creates a pyramid with a flat top.



1 Top radius

2 Height

22. Q

22.1 QCCLOSE command

Closes the **Calculator** panel.

Shape Lite Pro Mechanical BIM

22.1.1 Description

Closes the **Calculator** panel to hide it from the current workspace. If the **Calculator** panel is stacked when you close it, the **Calculator** tab or icon is removed from the stack.

22.2 QDIM command

Quickly creates a series of dimensions from selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

22.2.1 Method

The command is used for quickly generating a series of baseline or continued dimensions for each selected entity.

Select the geometry to dimension, then specify the dimension line position.

22.2.2 Options within the command

Continuous
Creates a succession of continuing dimensions where the lines are all positioned end-to-end on the same line.

Staggered
Creates a set of staggered dimensions where the linear dimension lines are spaced apart by a fixed distance.

Baseline
Creates a set of stacked dimensions starting from the same baseline.

Ordinate
Creates a set of ordinate dimensions with a single extension line with an X or Y value for each feature.

Radius
Creates radial dimensions and shows the radius value for the selected arcs and circles.

Diameter
Shows the diameter value of the selected circles or arcs.

datumPoint
Sets a new reference point for baseline and ordinate dimensions.

Edit
Adds/removes dimension points to/from the computed point set.

seTtings
Sets the entity snap priority.

EndPoint
Sets the entity snap priority to endpoint.

Intersection
Sets the entity snap priority to intersection.

22.3 QLATTACH command (Express Tools)

Associates a leader to an annotation entity.

Shape Lite Pro Mechanical BIM

Icon: 

22.3.1 Method

Select a leader entity, then select an MText, Tolerance or Block entity.

22.4 QLATTACHSET command (Express Tools)

Associates selected leaders with their annotation.

Shape Lite Pro Mechanical BIM

Icon: 

22.4.1 Method

Select leader entities. The number of unlocked leaders and attached annotations displays in the Command line.

22.5 QLDETACHSET command (Express Tools)

Disassociates selected leaders from an annotation.

Shape Lite Pro Mechanical BIM

Icon: 

22.5.1 Method

Select leader entities. The number of unlocked leaders and annotation detached displays in the Command line.

22.6 QLEADER command

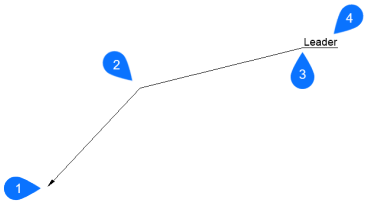
Draws leaders through a dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

22.6.1 Description

Draws leaders with a variety of annotation types and allows you to specify the properties through a dialog box.



1 Start point

2 Next point

3 To point

4 Annotation

22.6.2 Options within the command

Settings

The **QLeader Settings** dialog box pops up. Several settings can be specified there. See the related article **QLeader Settings dialog box**.

Text width

Specifies the text width for the text bounding box. Enter a value or 0 if no limit on the width needs to be set.

Note: When the length of the text exceeds the bounding box width, the text flows automatically to the next line.

22.7 QNEW command

Starts a new drawing based on the default template file.

Shape Lite Pro Mechanical BIM

Icon:

Alias: N

22.7.1 Description

Opens a new document tab based on the default template and user profile (short for "quick new").

22.8 QPRINT command

Prints the drawing using the default print configuration.

Shape Lite Pro Mechanical BIM

Icon:

22.8.1 Description

Allows printing a drawing without displaying the **Print** dialog box (short for "quick print").

If there is no printer added as a default printer a warning message will appear.

You will want to configure the drawing's print properties ahead of time with the PAGESETUP command.

22.9 QQUIT command (Express Tools)

Quickly closes all drawings and exits.

Shape Lite Pro Mechanical BIM

Icon: 

22.9.1 Method

If all drawings have been saved before, the command exits BricsCAD without displaying a dialog box. If there are unsaved files that are still open, you are asked if you want to save the drawings. For each unsaved drawing, a dialog box pops up and you can decide to save the drawing or not.

22.10 QRTEXT command

Creates an MText and displays it as a QR code.

Shape Lite Pro Mechanical BIM

Icon: 

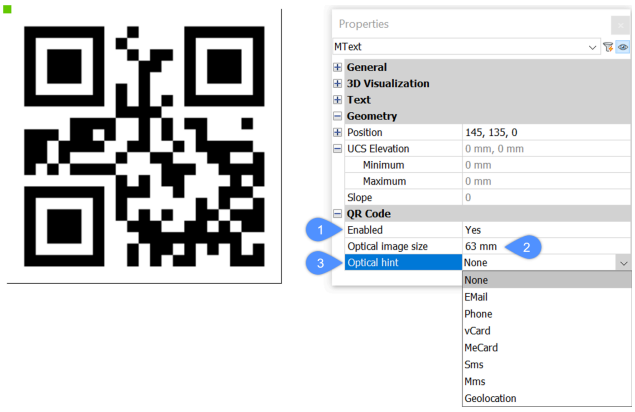
22.10.1 Description

Creates an MText which is represented as QR code by default.

Note:

- The QR code for a text is represented as a raster image.
- You can use the **Properties** panel to switch every text or mtext to QR code by setting the **Enabled** option to **Yes** (1), editing the **Optical image size** (2) and controlling the type of QR code by changing the **Optical hint** option (3).

  The **Optical hint** option allows you to specify 7 types of hints for QR code scanner: EMail, Phone, vCard, MeCard, Sms, Mms, Geolocation.



22.10.2 Options within the command

The Command line options are identical to the Command line options in the MTEXT command. See the MTEXT command article for details.

22.11 QSAVE command

Saves the drawing immediately.

Shape Lite Pro Mechanical BIM

Icon:

22.11.1 Description

Saves the drawing without displaying the **Save** dialog box (short for "quick save").

22.11.2 Method

If the drawing file has already been saved at least once, no dialog box pops up and the drawing is saved. To save the drawing by another name, use the SAVEAS command.

When the drawing has never been saved before or when the drawing is opened in read-only mode, the **Save Drawing As** dialog box pops up.

22.12 QSELECT command

Opens the **Properties** panel in **Quick Select** mode.

Shape Lite Pro Mechanical BIM

Icon:

22.12.1 Description

Opens the **Properties** panel in **Quick Select** mode to display it in the current workspace. The **Properties** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Properties** panel can be either floating, docked or stacked.

22.13 QRLEADER command

Creates a leader and displays its text as a QR code.

Shape Lite Pro Mechanical BIM

Icon:

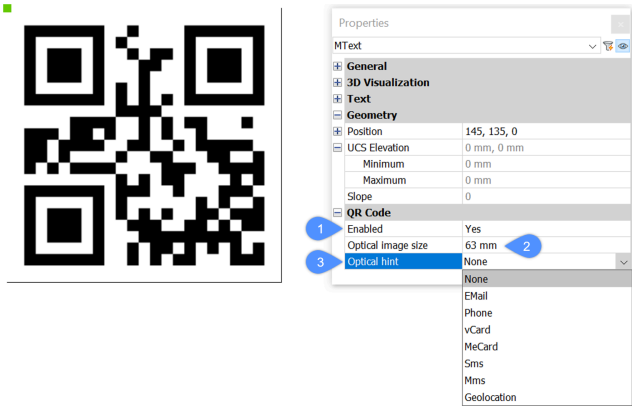
22.13.1 Description

Creates a leader with the text represented as QR code by default.

Note:

- The QR code for a text is represented as a raster image.
- You can use the **Properties** panel to switch every text or mtext to QR code by setting the **Enabled** option to **Yes** (1), editing the **Optical image size** (2) and controlling the type of QR code by changing the **Optical hint** option (3).

  The **Optical hint** option allows you to specify 7 hints for QR code scanner: EMail, Phone, vCard, MeCard, Sms, Mms, Geolocation.



22.13.2 Options within the command

The Command line options are identical to the Command line options in the LEADER command. See the LEADER command article for details.

22.14 QTEXT command

Toggles the QTEXTMODE system variable.

Shape Lite Pro Mechanical BIM

Alias: QT

22.14.1 Description

Toggles the QTEXTMODE system variable to display text entities as rectangles or text. You must use REGEN or REGENALL to see the change.

- On: turns on the QTEXTMODE system variable.
- Off: turns off the QTEXTMODE system variable.

- Toggle: changes the QTEXTMODE system variable to the opposite of the current setting.

22.15 QUADRANT command

Toggles the **Quadrant** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



22.15.1 Description

Toggles the **Quadrant** entity snap to enable or disable snapping to quadrant. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

22.16 QUICKCALC command

Opens the **Calculator** panel.

Shape Lite Pro Mechanical BIM

Icon: 

22.16.1 Description

Opens the **Calculator** panel to display it in the current workspace. The **Calculator** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Calculator** panel can be either floating, docked or stacked.

The **Calculator** panel provides a way to perform calculation inside BricsCAD. It has geometric functions, unit conversions, variables, and the basic features of a scientific calculator.



1 Get coordinates

2 Distance between two points

3 Angle of line defined by two points

4 Intersection of two lines defined by four points

5 Clear history

6 History panel

7 Input area

8 Paste value to command line

9 Scientific

10 Units conversion

11 Variables

22.16.2 Get coordinates

Retrieves coordinates of a picked point.

22.16.3 Distance between two points

Retrieves the distance between two picked points.

22.16.4 Angle of line defined by two points

Retrieves the angle of line defined by two picked points.

22.16.5 Intersection of two lines defined by four points

Retrieves the coordinates of the intersection of two lines defined by four picked points.

22.16.6 Clear history

Clears the history panel.

22.16.7 History panel

Displays all calculations.

22.16.8 Input area

Displays the calculations input and the retrieved geometrical functions.

22.16.9 Paste value to command line

Copies the value from the input area and pastes it to the Command line.

22.16.10 Scientific

Displays the scientific calculator functions.

22.16.11 Units conversion

Performs units conversions. You can choose from the drop-down lists the parameters.



1 Return conversion to input area

Return conversion to input area
Copies the converted value and pastes it to the input area.

22.16.12 Variables

Supports variables that can either be constants (coordinates/vectors, real numbers, and integers) or functions.



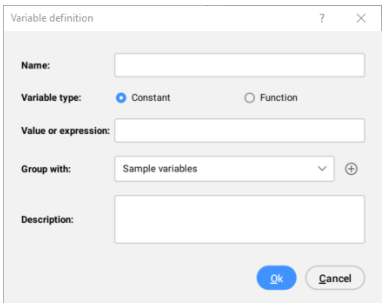
1 New variable

2 Edit variable

3 Delete

4 Return variable to input area

New variable...
Opens the **Variable definition** dialog box that allows you to define a new variable.



Note: User variables are stored in the calvar.xml file. The default folder for this file is *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*

Edit variable...
Edits the selected variable.

Delete
Removes the selected variable.

Return variable to input area
Copies the variable and pastes it to the input area.

Note: Double-click a variable to retrieve it to the input area.

Right-click a variable or a category to display a context menu with additional options:

New Category...
Opens the **Category definition** dialog box that allows you to define a new category.

22.16.13 Custom functions that can be used in Calculator expression

cur
Obtains a point by cursor. The **cur** function sets the value of the LASTPOINT system variable.
Sample:

- `cur()`
- `cur+2*[3,4]`

end, mid, cen, nod, qua, ins, per, nea
These snap modes are used as parts of arithmetic expressions. BricsCAD prompts you to select an entity and returns the coordinate of the appropriate snap point. Snap modes set the value of the LASTPOINT system variable.
Sample:

- `(end+mid)/2`

dist
Obtains the distance between two points.
Sample:

- `dist(cur,end)`

vec
Obtains the vector from two points.
Sample:

- `vec(end,mid)`

vec1
Obtains the unit vector from two points.
Sample:

- `vec1(cen,end)`

dee
Obtains the distance between two points. It is a shortcut for "dist(end,end)".
Sample:

- `dee`
- `dee()`

ill
Obtains the intersection of two lines defined by four endpoints.
Sample:

- `ill(end,end,cen,mid`

ille
It is a shortcut for "ill(ene,end,ene,end)".
Sample:

- `ille`
- `ille()`

mee
Obtains the midpoint between two endpoints. It is a shortcut for "(end+end)/2".
Sample:

- `mee`
- `mee()`

rad
Obtains the radius of a selected circle, arc, or polyline arc.
Sample:

- `rad`
- `rad()`

nor - without arguments
Obtains the unit normal vector of the selected object in UCS. Only a circle, arc, or polyline is accepted.
Sample:

- `nor`
- `nor()`

nor - with one argument
Obtains the unit normal to a 2D vector. The argument is treated as a 2D vector.
Sample:

- `nor(cur)`

- `nor([1,1])`

nor - with two arguments

Obtains the unit normal vector to two 2D endpoints. The Z-coordinate of the points is simply ignored.

Sample:

- `nor(cur,cur)`

nor - with three arguments

Obtains the unit normal vector by three 3D points.

Sample:

- `nor(cur,cur,cur)`

nee

It is a shortcut for "nor(end,end)".

vee

It is a shortcut for "vec(end,end)".

vee1

It is a shortcut for "vec1(end,end)".

getvar

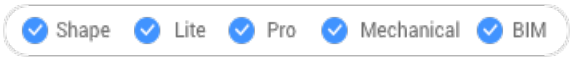
Returns the value of the BricsCAD system variables. Only real, integer or point system variables accepted as argument.

Sample:

- `getvar(UCSXDIR)`
- `getvar(textsize)`

22.17 QUIT command

Closes all open drawings and exits BricsCAD; offers the opportunity to save unsaved drawings.



Icon: 

Alias: EXIT

22.17.1 Method

If all drawings have been saved before, the command exits BricsCAD without displaying a dialog box.

If there are unsaved files that are still open, you are asked if you want to save the drawings. For each unsaved drawing, a dialog box pops up and you can decide to save the drawing or not.

23. R

23.1 RAY command

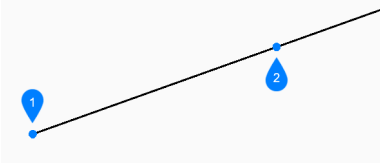
Creates rays.

Shape Lite Pro Mechanical BIM

Icon:

23.1.1 Description

Creates a ray (semi-infinite line) from a combination of options including point, direction and angle.



1 Start of ray

2 Direction

23.1.2 Method

This command has 6 methods to begin creating a ray:

- Start of ray
- Horizontal
- Vertical
- Angle
- Bisect
- Parallel

23.1.3 Options within the command

Start of ray
Allows you to begin creating a ray by specifying the starting point of the ray.

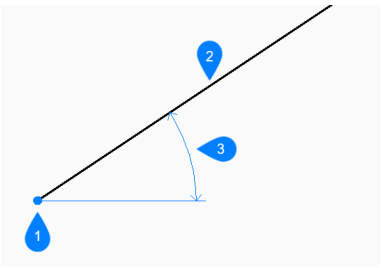
Direction
Specifies the direction of the ray from the starting point.

Note: You can continue adding unlimited rays until you press **Enter** key to end the command.

Horizontal
Creates a ray horizontal to the x-axis.

Vertical
Creates a ray parallel to the y-axis.

Angle
Creates a ray based on a specified angle.

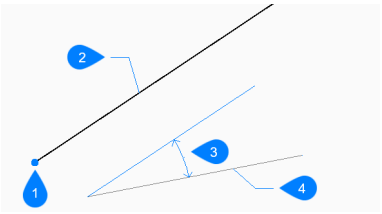


1 Location

2 Ray

3 Angle

Reference
Specifies the angle to place the ray relative to the selected entity.



1 Location

2 Ray

3 Angle

4 Reference entity

Location
Specifies the starting point of the horizontal, vertical, or angle ray.

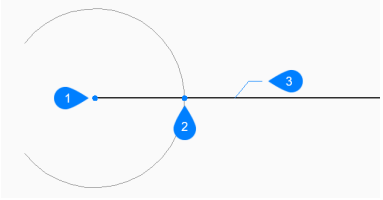
Bisect
Creates a ray which bisects the angle between two imaginary lines.

Set vertex point
Specifies the start point of the ray.

Entity
Allows you to select a line, arc, or polyline segment to bisect.

When you select a line or polyline segment, the command draws the xline perpendicular to the segment's midpoint. When you select an arc or polyarc, the command draws the xline perpendicular to the arc's center and midpoint.

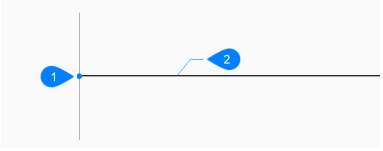
Note: This option works with splined polylines, but not with splined entities.



1 Perpendicular to arc center point

2 Perpendicular to arc midpoint

3 Ray

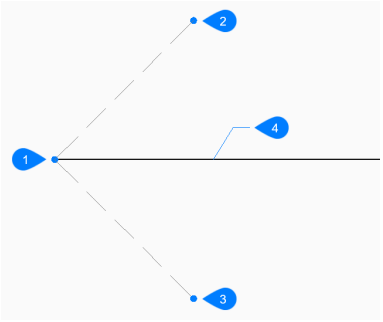


1 Perpendicular to line midpoint

2 Ray

Bisection angle start point
Specifies a point to define the first imaginary line. The vertex is used as the other point.

Bisection angle end point
Specifies a point to define the second imaginary line. The vertex is used as the other point.



1 Vertex point

2 Bisection angle start point

3 Bisection angle end point

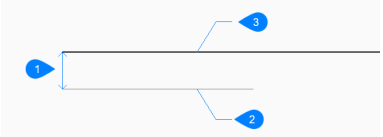
4 Ray

Parallel
Creates a ray parallel to a line or polyline segment.

Set offset distance for parallel infinite ray
Specifies the offset distance for the ray.

Select entity for parallel infinite ray
Allows you to select a line or polyline segment from which to offset the ray.

Side for parallel infinite ray
Specifies the side on which to place the ray. The ray's start point is parallel to the line's start point.



1 Offset distance

2 Line segment

3 Ray

Through point
Specifies the point through which to draw the ray. This point is the ray's start point.



1 Through point

2 Line segment

3 Ray

23.2 REASSOCAPP command

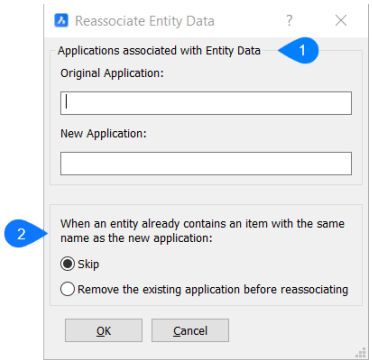
Opens the **Reassociate entity data** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

23.2.1 Description

Opens the **Reassociate entity data** dialog box to associate extended entity data with a specific application.



1 Applications associated with Entity Data

2 Options

23.2.2 Applications associated with Entity Data

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.

23.2.3 Options

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.

23.3 RECOVER command

Opens the **Open drawing** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

23.3.1 Description

Opens the **Open drawing** dialog box to select a damaged DWG, DWT, or DXF file from which to recover data.

23.4 RECOVERALL command

Opens the **Open drawing** dialog box.

Shape Lite Pro Mechanical BIM

23.4.1 Description

Opens the **Open drawing** dialog box to select a damaged DWG, DWT, or DXF file from which to recover data. In addition to the selected file, BricsCAD will attempt to recover data from all nested external references.

23.5 RECSCRIPT command

Records keystrokes in the current drawing and saves them to an SCR script file.

Shape Lite Pro Mechanical BIM

Icon: 

23.5.1 Description

Opens the **Record script** dialog box where you have to choose an SCR file. After choosing **Save** in the dialog box, the commands and pick points you specify in the drawing editor are recorded in the SCR file until you run the STOPSCRIPT command.

Note: To run the recorded script, launch the SCRIPT command.

23.6 RECTANG command

Creates a polyline in the shape of a rectangle.

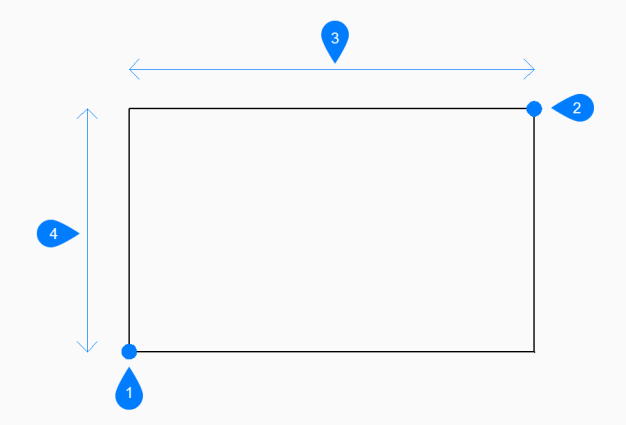
Shape Lite Pro Mechanical BIM

Icon:

Alias: REC, RECT, RECTANGLE

23.6.1 Description

Creates a closed, four-sided rectangular polyline in the shape of a rectangle, square, and variants with chamfers or fillets.



1 First corner

2 Second corner

3 Length

4 Width

23.6.2 Method

This command has 5 methods to begin creating a rectangle:

- Select first corner of rectangle
- Rotated
- Square
- Area
- Dimensions

23.6.3 Options within the command

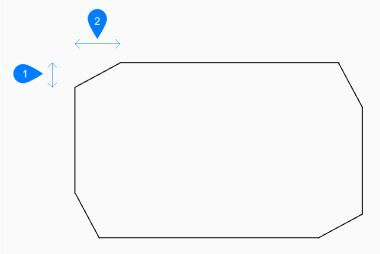
Select first corner of rectangle

Allows you to begin creating a rectangle by specifying the first corner of a rectangle.

Other corner of rectangle

Specifies the opposite corner of the rectangle. The rectangle is drawn parallel to the X and Y axes.

Chamfer
Specifies the first and second chamfer distance to use for all rectangles.



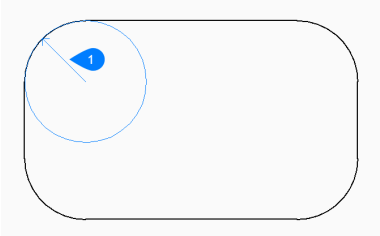
1 First chamfer distance

2 Second chamfer distance

turn chamfer Off
Turns **Off** the **Chamfer** option.

use Default setting
Uses the default settings for chamfer, specified by the CHAMFERA, CHAMFERB, CHAMFERC, CHAMFERD, CHAMMODE system variables.

Fillet
Specifies the fillet distance to use for all rectangles.

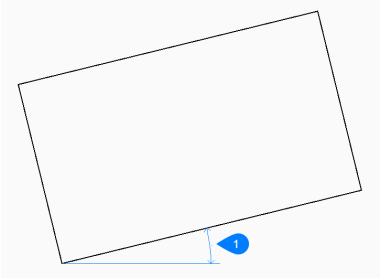


1 Fillet radius

turn fillet Off
Turns **Off** the **Fillet** option.

use Default setting
Uses the default setting for fillet, specified by the FILLETRAD system variable.

Rotated
Creates a rectangle by specifying two corners and the rotation angle.



1 Rotation angle

Square
Begins creating a rectangle by specifying a corner of a square.

Second corner of square
Specifies a corner along the same side of the square to define its length and angle.

Elevation
Specifies the height above the XY-plane to use for all rectangles.

use Default
Uses the default setting for elevation, specified by the ELEVATION system variable.

Thickness
Specifies the thickness of the rectangle line segments. All segments have the same thickness.

Width of line
Specifies the width of the rectangle line segments. All segments have the same width.

Area
Creates a rectangle by specifying its area.

Length
Calculates the rectangle dimension based on length.

Width
Calculates the rectangle dimension based on width.

Dimensions
Creates a rectangle by specifying its length and width.

23.7 REDEFINE command

Activates commands that were deactivated with the UNDEFINE command.

Shape Lite Pro Mechanical BIM

23.7.1 Description

Reactivates commands that were deactivated with the UNDEFINE command by entering the name of the undefined command.

23.8 REDIR command (Express Tools)

Redefines hard-coded paths in external references, images, shapes, styles and rtexts.

Shape Lite Pro Mechanical BIM

Icon: 

23.8.1 Method

1 Enter the old directory.

Note: Type * to redefine all.

2 Specify the new directory.

23.8.2 Options within the command

options
Opens the **REDIR Settings** dialog box that allows you to select which element types you want to replace the directories from.

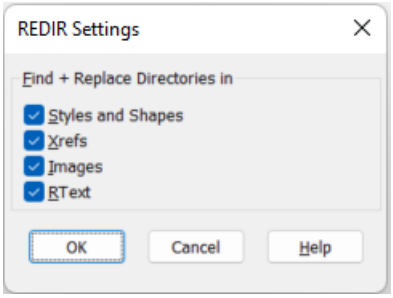
23.9 REDIRMODE command (Express Tools)

Sets the type of entities to find and replace the directories from.

23.9.1 Method

Opens the **REDIR Settings** dialog box.

The **REDIR Settings** dialog box defines which types of entities to include for the REDIR command.



Note: At least one option must be selected.

23.10 -REDIRMODE command (Express Tools)

Sets the type of entities you want to find and replace the directories from via the Command line.

Shape Lite Pro Mechanical BIM

23.10.1 Method

Specify the type of entities to be included when using the REDIR command, separated by comma. There is no need to type the complete word, for example, s, x, i, r are valid entries.

Note: Type * to specify all type of entities.

23.11 REDO command

Reverses the action of the previous U or UNDO command.

Shape Lite Pro Mechanical BIM

Icon:

23.11.1 Description

Reverses the action of the previous U or UNDO command to restore entities to their state prior to the U or UNDO operation. The REDO command only works immediately following the U or UNDO commands.

This command displays no prompts and has no options.

23.12 REDRAW command

Redraws entities in the current viewport.

Shape Lite Pro Mechanical BIM

Icon:

Alias: R

23.12.1 Description

Redraws entities in the current viewport to remove graphic debris such as blip or drag marks.

This command displays no prompts and has no options.

23.13 REDRAWALL command

Redraws entities in all viewports.

Shape Lite Pro Mechanical BIM

Icon:

Alias: RA

23.13.1 Description

Redraw entities in all viewports to remove graphic debris such as blip or drag marks.

This command displays no prompts and has no options.

23.14 REDSDKINFO command

Displays rendering related hardware and driver specifications in the Command line.

Shape Lite Pro Mechanical BIM

23.14.1 Description

Prints a report in the command window containing information about the graphics capabilities of your computer.

Press F2 to open the prompt history window.

23.15 REFCLOSE command

Closes the reference drawing editor.

Shape Lite Pro Mechanical BIM

Icons:

23.15.1 Description

Closes the reference drawing editor, used to edit externally-referenced files in the current drawing session.

Note: This command can only be used after the REFEDIT command has commenced.

23.15.2 Method

There are 2 methods to close the reference drawing editor:

- Save
- Discard

23.15.3 Options within the command

Save
Saves the Reference drawing including the changes.

Discard
Saves the Reference drawing without saving the changes.

23.16 REFEDIT command

Edits block references and externally referenced drawings.

Shape Lite Pro Mechanical BIM

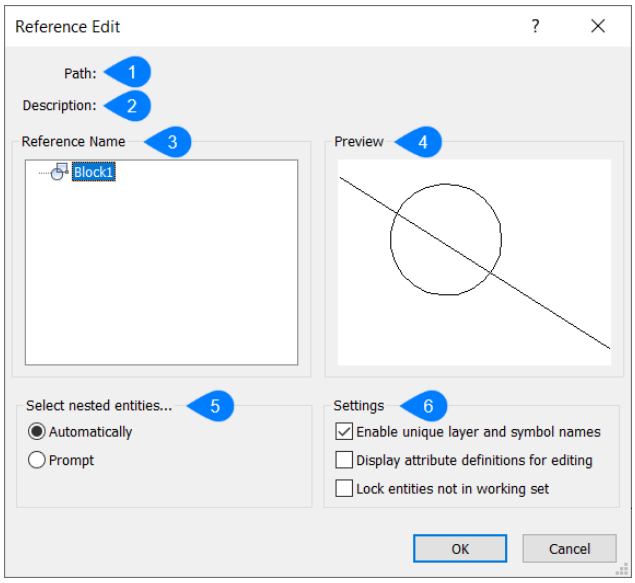
Icon: 

23.16.1 Description

Edits the selected block reference or externally referenced drawing. The **Reference Edit** dialog box displays after you select a block. You can use the REFCLOSE command or the **Ref Edit** toolbar to end the reference editing session. The XFADECTL system variable sets the fading for the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted. If the XEDIT system variable of the selected external reference is 0 or OFF, a dialog box displays a message that you cannot edit the selected xref block.

Note: Only one reference can be edited at a time.

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

23.16.2 Path

Displays the name of the path.

23.16.3 Description

Displays the description of the path.

23.16.4 Reference Name

Lists the name of the block and nested blocks, if any.

23.16.5 Preview

Displays a preview of the current reference.

23.16.6 Select nested entities

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.

23.16.7 Settings

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 entities not in working set
Toggles locking of entities not being edited.

23.17 -REFEDIT command

Edits block references and externally referenced drawings.

Shape Lite Pro Mechanical BIM

23.17.1 Description

Edits the selected block or externally-referenced drawing. The XFADECTL system variable sets the fading the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted. Use the REFCLOSE command or the **Ref Edit** toolbar to end reference editing session.

Note: Only one reference can be edited at a time.

23.17.2 Options within the command

Select nesting level
Specifies a nested reference to edit.

Ok
Edits the selected reference.

Next
Edits reference on a deeper nesting level.

Note: This option will keep repeating until 0 is entered.

Enter entity selection method
Specifies how nested references are selected.

All
Selects all nested entities automatically.

Nested
Selects specific nested entities.

inside Window
Selects all nested entities inside a rectangular window, specified by picking 2 opposite corners.

Crossing window
Selects all nested entities crossing a rectangular window, specified by picking 2 opposite corners.

Window Polygon
Selects all nested entities inside a polygonal window, specified by picking points.

Crossing Polygon
Selects all nested entities crossing a polygonal window, specified by picking points.

Add
Adds entities to the selection.

Remove
Removes entities from selection.

Undo
Undoes the last step of selecting entities.

Display attribute definitions
Toggles the inclusions of attributes on or off.

Yes
Attribute definitions can be edited, all attribute values are visible.

No
Attributes are unavailable for editing.

Note: Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on.

23.18 REFERENCECURVES command

Creates reference geometry to automatically align a block during insertion.

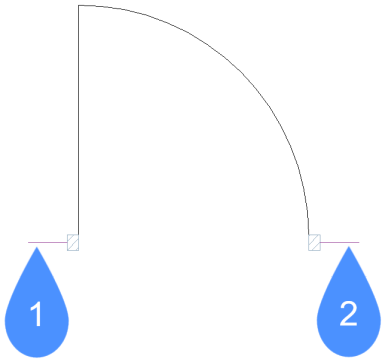
Shape Lite Pro Mechanical BIM

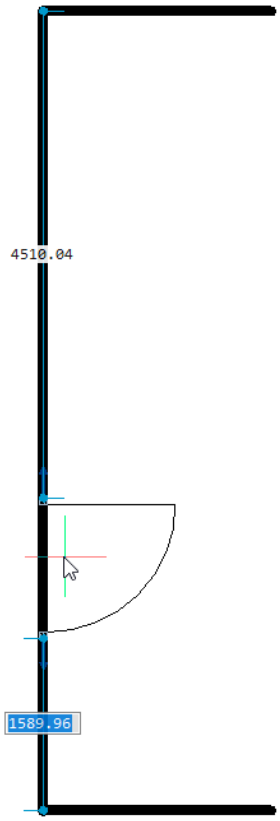
Icon: 

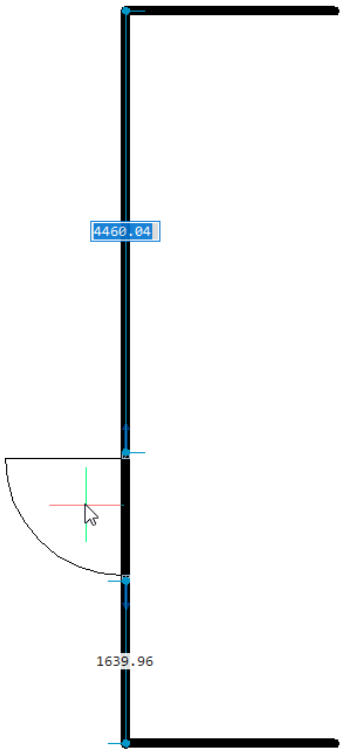
23.18.1 Method

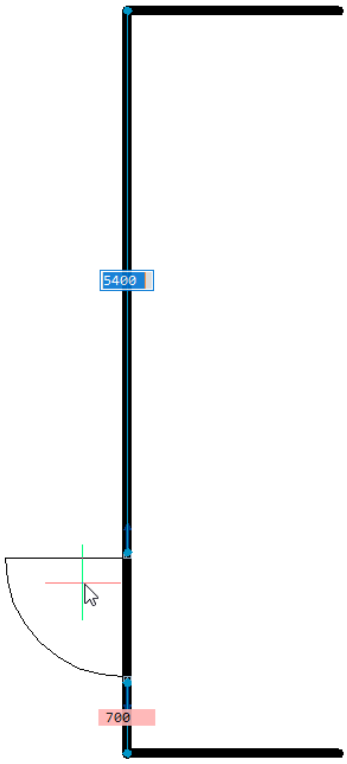
The selected entities which you want to use as a reference to align the inserted block will be moved in a new created REFERENCE_CURVES layer (if it doesn't already exist).

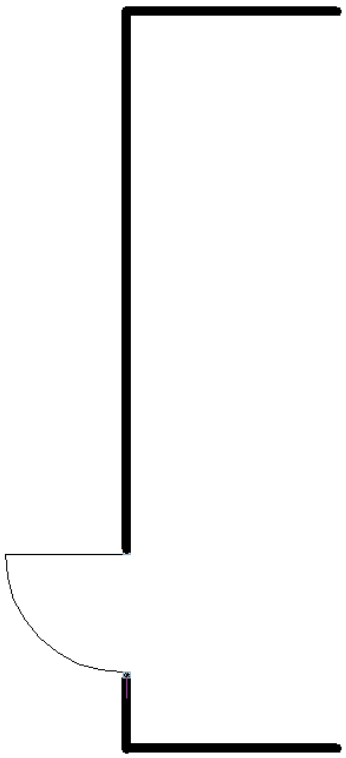
With reference curves ((1) and (2)) specified in a drawing or block definition, you can automatically align the block or drawing to relevant geometry when you insert it. The number of reference curves and the distance between them determines with which geometry it can align. As the cursor approaches 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.


1
2

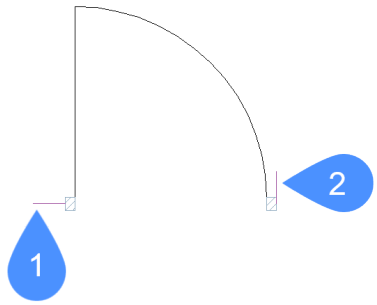

4510.04
1589.96

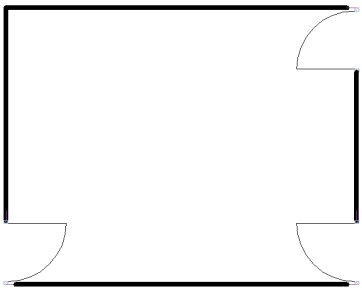

4460.04
1639.96


5400
700



You can also use reference curves to automatically align with corners. The following example includes a parallel reference curves (1) and a corner reference curve (2) enabling the door block to align with geometry that matches the parallel line and the perpendicular line.

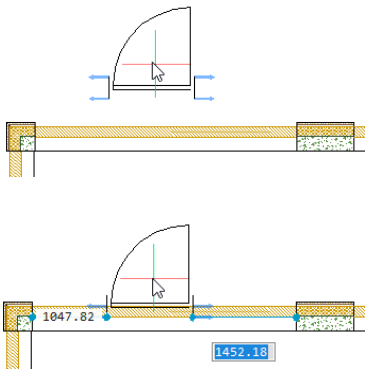




23.18.2 Options within the command

Parametrize

Parametrizes the reference curves, so that the resulting parametric block can be inserted in a "fuzzy" mode. For example, you can parametrize the four parallel reference curves in a simple door example. Then you can insert in a guided way the resulting parametric door block into walls with different thicknesses.



Accept

Creates reference curves without parametrizing.

23.19 REFSET command

Adds or removes entities from the reference (block or xref) being edited.

Shape Lite Pro Mechanical BIM

Icons: 

23.19.1 Description

Transfers objects between the RefEdit working set and host drawing.

Note: This command can be used only after the REFEDIT command has commenced.

23.19.2 Options within the command

Add

Adds entities from the drawing to the reference being edited.

Remove

Removes entities from the reference being edited; removed entities are placed in the drawing. Entities not in the reference are shown in gray.

23.20 REGEN command

Regenerates entities in the current viewport.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: RE

23.20.1 Description

Regenerates entities in the current viewport for optimal display and selection performance.

REGEN does the following:

- Recalculates the location and visibility of all entities in the current viewport.
- Reindexes the drawing database.
- Updates the area available for zooming and panning.

This command displays no prompts and has no options.

23.21 REGENALL command

Regenerates entities in all viewports.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: REA

23.21.1 Description

Regenerates entities in all viewports for optimal display and selection performance.

REGENALL does the following:

- Recalculates the location and visibility of all entities.
- Reindexes the drawing database.
- Updates the area available for zooming and panning.

This command displays no prompts and has no options.

23.22 REGENAUTO command

Toggles the REGENMODE system variable.

Shape Lite Pro Mechanical BIM

23.22.1 Description

Toggles the REGENMODE system variable to specify if the display automatically regenerates when needed. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'REGENMODE.

- On: turns on the REGENMODE system variable.
- Off: turns off the REGENMODE system variable.
- Toggle: changes the REGENMODE system variable to the opposite of the current setting.

23.23 REGION command

Creates regions.

Shape Lite Pro Mechanical BIM

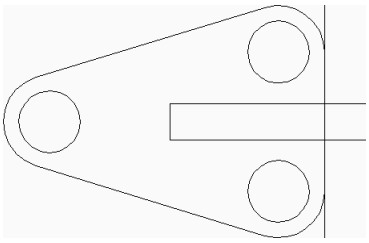
Icon:

Alias: REG

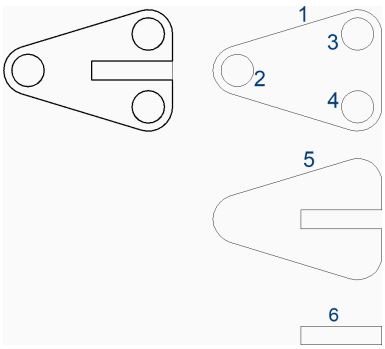
23.23.1 Description

Creates regions from closed entities or sets of entities that enclose a space.

Original entities:



Regions (6):



23.23.2 Method

Select closed entities or sets of entities that enclose a space. Each closed entity or set of entities produces a region.

You can continue creating regions until you press **Enter** key to end the command.

Depending on the current value of the DELOBJ system variable the original geometry is deleted or retained.

23.24 REINIT command

Reloads the alias (PGP) file.

Shape Lite Pro Mechanical BIM

Alias: RI

23.24.1 Description

Reloads the alias (PGP) file after it has been edited externally. A **BricsCAD** dialog box confirms if you want to reload the file.

23.25 REMOVESKETCHFEATURE command

Removes all sketch features at once but keeps the corresponding 3D solid.

Shape Lite Pro Mechanical BIM

23.25.1 Method

Specify if you want to remove or not all existing sketch features created when the CREATESKETCHFEATURE system variable was enabled.

Note: The corresponding 3D solids will remain.

23.26 RENAME command

Renames named entities through the **Drawing explorer** dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDRENAME, REN

23.26.1 Description

Opens the **Drawing Explorer** dialog box on the last used tab or category to manage definitions and reference content that is used in the drawing. Rename can be accessed through the context menu.

Note: To change names at the Command line, use the -RENAME command.

You cannot change the names of attached files, such as images and external references, nor the following names:

- Layer "0"
- Linetype "ByBlock", "ByLayer", or "Continuous"
- Text style "Standard"
- Dimension style "Standard"
- Multiline style "Standard"
- Visual style "2DWireframe"
- Material "Global"
- Page layouts that have * as a prefix and suffix, such as *Model*
- View detail style "Metric50"
- View section style "Metric 50"

23.27 -RENAME command

Renames entities in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -REN

23.27.1 Options within the command

Block
Renames blocks.

Dimension style
Renames dimension styles.

LAyer
Renames layers.

LineType
Renames linetypes.

text Style
Renames text styles.

Table style
Renames table styles.

Ucs
Renames user-defined coordinate systems.

Vlew
Renames views.

ViewPort
Renames viewport configurations.

23.28 RENDER command

Opens the **Render** dialog box.

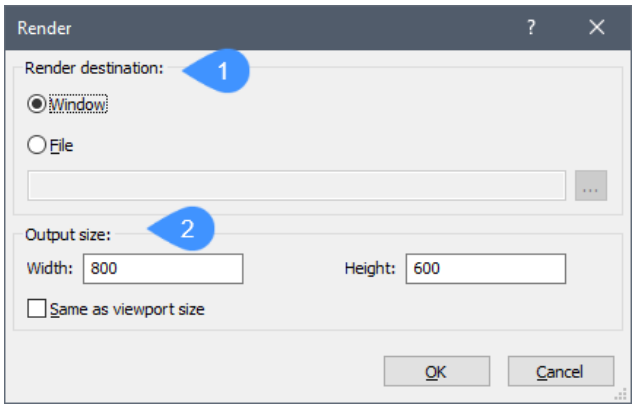
Shape Lite Pro Mechanical BIM

Icon: 

Alias: RR

23.28.1 Description

Opens the **Render** dialog box to create a photo-realistic rendering of the current drawing.



1 Render destination

2 Output size

23.28.2 Render destination

Specifies where the render will appear.

Window
Generates the render in a new window.

File
Saves the render to a file. Selecting the **Browse** button opens the **Render output file** dialog box.

23.28.3 Output size

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.

23.29 -RENDER command

Renders 3D models.

Shape Lite Pro Mechanical BIM

23.29.1 Description

Generates photo-realistic renderings of 3D models in a file or window and applies materials and lights if available. It works from the Command line.

23.29.2 Options within the command

Render preset

Choose a predefined preset rendering style or choose **Other** to select a custom preset created with the RENDERPRESET command:

- Draft: renders 3D models without ray tracing; fastest rendering speed.
- Low: renders with ray tracing set to level 3 (low).
- Medium: renders with ray tracing set to level 5 (medium).
- High: renders with ray tracing set to level 7 (high).
- Presentation: renders with ray tracing set to level 9 (highest); slowest rendering speed.
- Other: specify the name of a user-defined rendering preset.

Render destination

Display the rendering in one of the following locations:

- File: saves the rendering as a BMP file in the folder defined by the DWGPREFIX system variable.
- Render window: displays the rendering in a separate window. The dimensions of the rendering are specified in pixels.

23.30 RENDERPRESETS command

Creates and edits properties for renderings through **Drawing Explorer** dialog box.

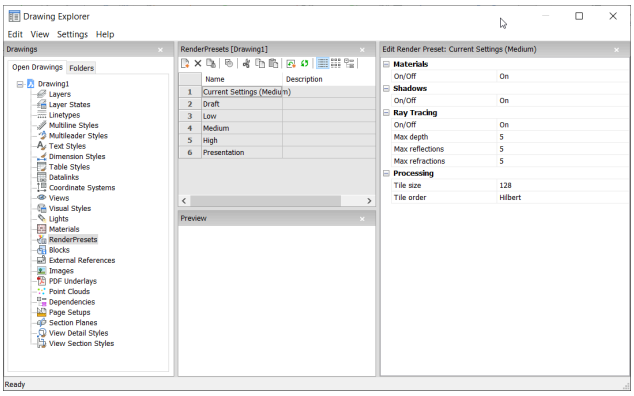
Shape Lite Pro Mechanical BIM

Icon:

Alias: ROPTIONS

23.30.1 Description

Opens the **Drawing Explorer** dialog box with **RenderPresets** category selected to view and modify render presets in the current drawing.



23.30.2 Options within the Edit Render Preset panel

Materials

Toggles the use of materials in a rendering.

Note: Materials are applied to layers and entities. See MATERIALS command.

On
Uses material definitions for rendering.

Note: This comes with the cost of a slower rendering process.

Off
Does not use materials.

Note: This comes with the cost of a less realistic rendering

Shadows
Toggles the use of shadows in the rendering.

On
Uses shadows for rendering.

Note: This comes with the cost of a slower rendering process.

Off
Does not use shadows.

Note: This comes with the cost of a less realistic rendering.

Ray Tracing
Tracks each ray of light from each light source as it travels through the scene and bounces off entities.

On
Uses ray tracing.

Note: This comes with the cost of a slower rendering process.

Off
Does not use ray tracing.

Note: This comes with the cost of a less realistic rendering.

Max Depth
Specifies the maximum total number of times light bounces and refracts.

Max Reflections
Specifies the maximum times that light beams bounce off entities.

Max Refractions
Specifies the maximum times that light refract through transparent entities.

Processing
Specifies how the rendered image is processed, applied to renderings sent to a separate window.

Tile Size
Sets the size of the tile, representing the area of the rendered image being processed.

Tile Order
Defines the order in which the tiles are generated.

- **Hilbert**: renders with a Hilbert curve, a continuous fractal space-filling; see http://en.wikipedia.org/wiki/Hilbert_curve for a description.

- **Spiral**: renders in a counterclockwise spiral from the center.
- **Left to right**: renders vertical columns, starting from the bottom left corner.
- **Right to left**: renders vertical columns, starting from the bottom right corner.
- **Top to bottom**: renders horizontal rows, starting at the top left corner.
- **Bottom to top**: renders horizontal rows, starting at the bottom left corner.

23.30.3 Context Menu Options

New
Creates additional Render Preset definitions into the drawing. Displays the **New Render Preset** dialog box.

Delete
Deletes Render Preset definitions from the drawing. The following Render Preset definitions cannot be deleted:

- **Draft**
- **Low**
- **Medium**
- **High**
- **Presentation**

Set current
Sets the selected Render Preset as current.

Note: The current render preset is used by the RENDER command or when printing using the shade plot type **Rendered** as defined in **Shade plot** setting in the PAGESETUP and the PRINT commands. The name of the current render preset displays between brackets: e.g. Current Settings (Medium).

Note: You can override the properties of a preset in the **Edit Render Preset** pane. When overrides exist, an asterisk displays in front of the current render preset name: e.g. Current Settings (*Medium).

Rename
Renames the Render Preset.

Note: Following Render Preset items cannot be renamed:

- **Draft**
- **Low**
- **Medium**
- **High**
- **Presentation**

Select All
Selects all Render Preset definitions.

Invert selection
Deselects the current selection and vice versa.

23.31 RENDERWINCLOSE command

Closes the Render window.

Shape Lite Pro Mechanical BIM

23.31.1 Description

Closes the Render window, which had displayed the result of the Render command.

23.32 REPORTPANELCLOSE command

Closes the **Report** panel.

Shape Lite Pro Mechanical BIM

23.32.1 Description

Closes the **Report** panel to hide it from the current workspace. If the **Report** panel is stacked when you close it, the Report tab or icon is removed from the stack.

23.33 REPORTPANELOPEN command

Opens the **Report** panel.

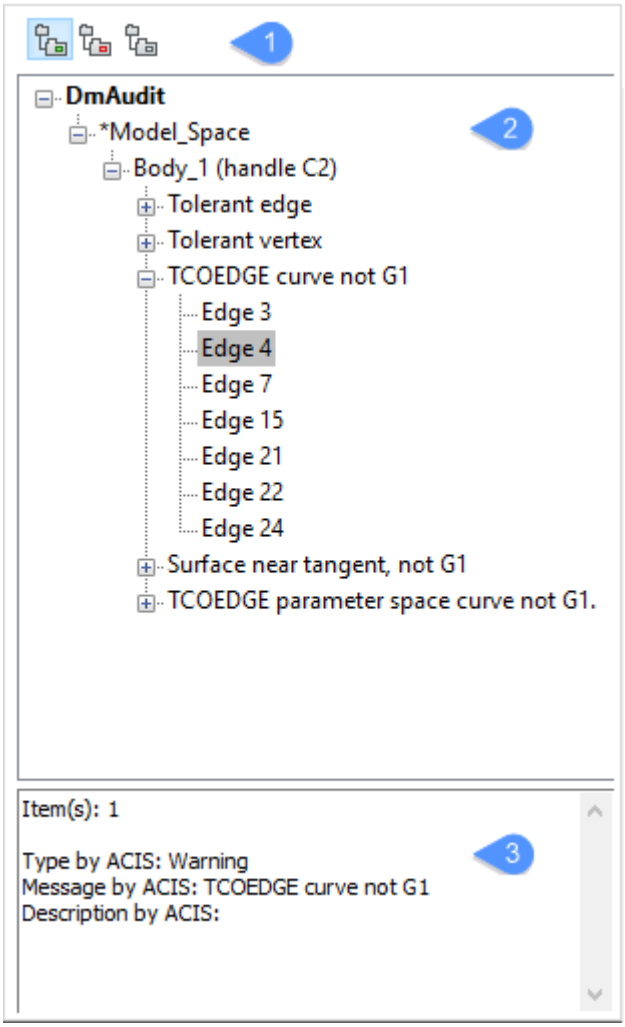
Shape Lite Pro Mechanical BIM

23.33.1 Description

Opens the **Report** panel to display it in the current workspace. The **Report** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Report** panel can be either floating, docked or stacked.

The **Report** panel offers feedback for relevant commands including DMAUDIT, DMSIMPLIFY, DMSIMPLIFYALL, 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

23.33.2 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

23.33.3 Report output

Displays command output.

23.33.4 Detailed items output

Shows detailed command output for selected item in the report output.

23.34 REPURLS command (Express Tools)

Finds and replaces previously attached URL addresses.

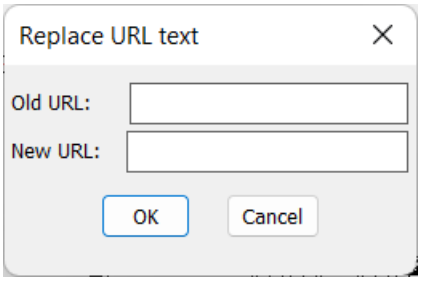
Shape Lite Pro Mechanical BIM

Icon:

23.34.1 Method

Select the entities. The **Replace URL text** dialog box opens.

The **Replace with URL text** dialog box allows you to replace a specified text string in a URL used in hyperlinks attached to the selected entities.



23.35 RESETASSOCVIEWS command

Removes associativity between 2D drawings and 3D models.

Shape Lite Pro Mechanical BIM

23.35.1 Description

Removes the associativity between 2D drawings and 3D models, which had been generated by commands like VIEWBASE. This can be used to “freeze” drawing layouts at certain stages throughout the 3D model development, such as for archiving plans.

23.36 RESETBLOCK command

Resets parametric blocks to default values.

Shape Lite Pro Mechanical BIM

23.36.1 Description

Parametric blocks can be manipulated by users interactively, and so this command sets the blocks back to their original condition.

23.37 RESUME command

Resumes paused scripts.

Shape Lite Pro Mechanical BIM

23.37.1 Description

Resumes scripts after they have been stopped with **Esc** key.

23.38 REVCLOUD command

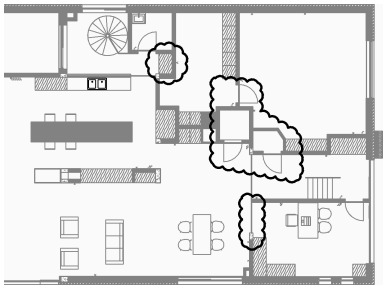
Creates a polyline in the shape of a revision cloud.

Shape Lite Pro Mechanical BIM

Icon:

23.38.1 Description

Creates a closed polyline in the shape of a revision cloud. Choose from a combination of options including rectangular, polygonal, freehand, and entity.



23.38.2 Options within the command

Arc length

Specifies the shortest and longest length for the arcs that form the cloud.

Specify minimum length of arc

Specifies the minimum length of the arcs.
The minimum arc length is stored in the REVCLOUDMINARCLENGTH system variable.

Specify maximum length of arc

Specifies the maximum length of the arcs.
The maximum arc length is stored in the REVCLOUDMAXARCLENGTH system variable.

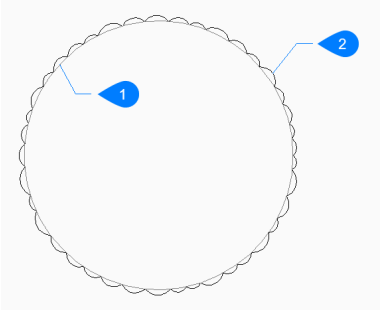


1 Minimum arc length

2 Maximum arc length

Entity

Creates a revision cloud by selecting an existing entity to convert to a revision cloud.



1 Original entity

2 Revision cloud

Note: Entities can be open or closed. The original entity is erased.

Rectangular

Creates a rectangular revision cloud by specifying the first and opposite corner of the rectangle. The revision cloud is drawn parallel to the x and y axes.

Polygonal

Creates a polygonal revision cloud by specifying the vertices of the polygon. You can continue adding unlimited vertices until you press **Enter** key to end the command.

Undo

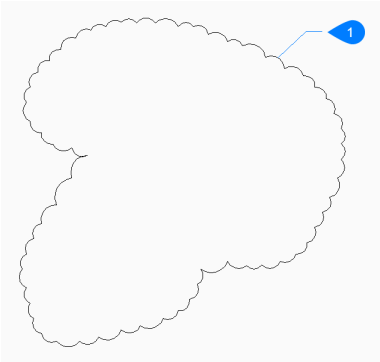
Undoes the last segment and continues drawing from the previous segment.

Freehand

Creates a freehand revision cloud.

Guide crosshairs along cloud path...

Allows you to drag the cursor to add revision cloud segments. Continues adding segments until you pass the cursor over the start point to create a closed revision cloud.



1 Closed revcloud

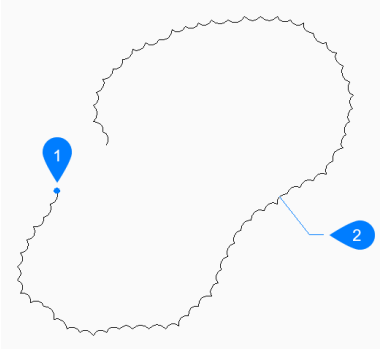
Press **Enter** key to stop adding revision cloud segments at anytime.

Reverse direction

Allows you to specify the direction of the revision cloud.

Yes

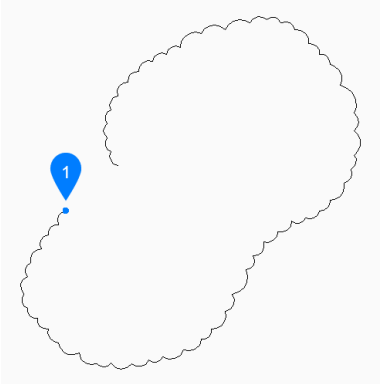
The direction of the arc segments is reversed.



1 Start point

2 Reversed

No
The direction of the arc segment is retained.

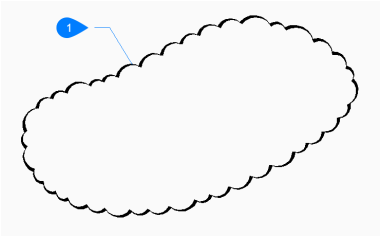


1 Start point

Style
Specifies the arc style to use for the revision cloud.

Normal
The polyline making up the cloud has a uniform width.

Calligraphy
The polyline has a varying width (tapered polyarcs) to simulate a calligraphic look.



1 Calligraphic style

Note: The arc style is stored in the REVCLOUDARCSTYLE system variable.

23.39 REVERT command (Express Tools)

Closes the current drawing without saving and reopens it.

Shape Lite Pro Mechanical BIM

Icon: 

23.39.1 Method

If there are unsaved changes in the current drawing, BricsCAD asks if you want to discard the changes.

23.40 REVOLVE command

Creates 3D solids or surfaces by revolving 2D entities about an axis.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: REV

23.40.1 Description

Revolves open or closed 2D entities, solid edges, faces of 3D solids, regions or closed boundaries into 3D solids or 3D surfaces.

Note:

- The SELECTIONPREVIEW system variable must be set to 2 or 3 to highlight faces.
- Depending on the value of the DELOBJ system variable, the source entities are either retained or deleted, or else you are prompted whether you want the entities deleted or not.

23.40.2 Method

There are two methods to revolve entities:

- Create 3D solids.
- Create surfaces.

23.40.3 Options within the command

MOde
Allows you to create either solids or surfaces.

SOlid
Creates a 3D solid.

Note: Only for **Solid** mode, when the CREATESKETCHFEATURE system variable is ON, sketch features for revolve are created in a dedicated BC_SKETCHES layer, which is not visible by default. The sketch is visible and editable as block references in the **Mechanical Browser** panel. Also, the sketch features for revolve are visible and editable as block references in the **Mechanical Browser** panel.

Note: The CREATESKETCHFEATURE system variable can also be controlled by pressing the **CreateSketchFeature** toggle button  in the ribbon.

SUrface
Creates a 3D surface.

Xaxis
Uses the X axis of the current coordinate system as the axis of rotation.

Yaxis
Uses the Y axis of the current coordinate system as the axis of rotation.

Zaxis
Uses the Z axis of the current coordinate system as the axis of rotation.

2Points
Specifies the start point and end point of the axis of revolution.

Note: If the resulting 3D entity is self-intersecting, an error will occur.

Object
Specifies the axis of rotation by selecting an axial entity of the object.

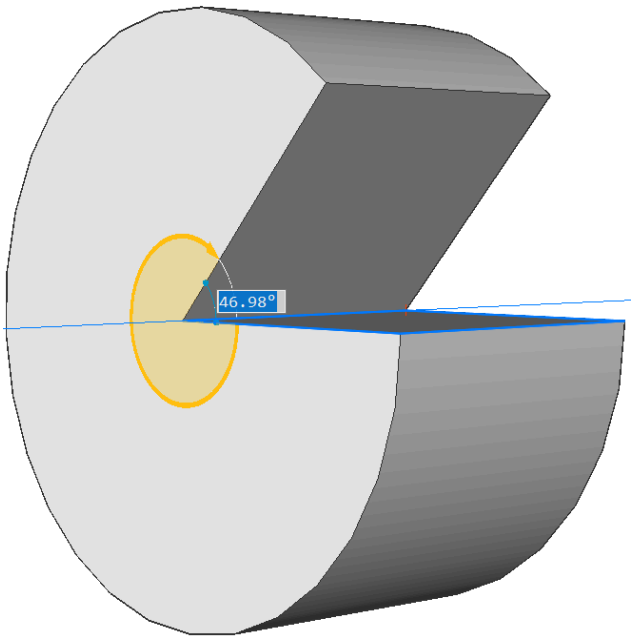
Last
Uses the last entered axis for rotating the entities.

View
Uses the current view as the axis for rotating the entities.

Note: The axis of rotation is parallel to the direction of view, passing through the picked point.

Angle of revolution
Specifies how far around the source entity is rotated about the axis.

Note: You can specify the angle of revolution dynamically, using the **Manipulator**, or typing a value.



Auto
Move the mouse to specify the angle direction.

The result depends on the extrusion direction and the value of the four **Extrude mode** system variables: EXTRUDEOUTSIDE, EXTRUDEINSIDE, INTERSECTEDENTITIES and UNITESURFACES.

Subtract
The 3D solid is subtracted from each interfering existing solid.

Create
Regardless of the revolving direction, a new 3D solid is created.

Unite
The new 3D solid is unified with each interfering existing solid.

Both sides
Revolves in both directions.

Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the **Ctrl** key during the dynamic display of the revolve to cycle the various options.



23.41 REVSURF command

Creates a 3D mesh surface by revolving a linear entity around a line (short for "revolved surface").

Shape Lite Pro Mechanical BIM

Icon:

Note: To create 3D surfaces and 3D solids by revolving a profile around an axis, use the REVOLVE command.

The SURFTAB1 system variable controls the number of segments of the revolution surface.

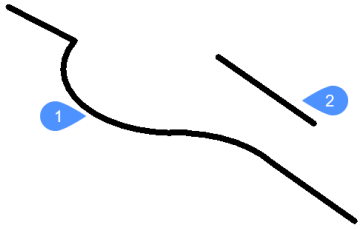
The SURFTAB2 system variable controls the number of segments of each arc segment in the revolved entity

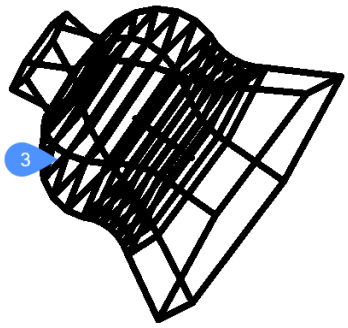
23.41.1 Method

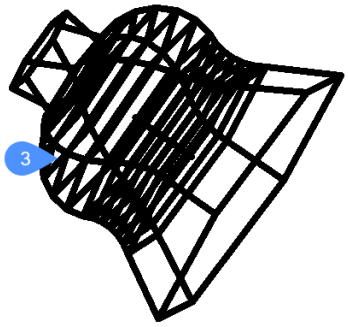
Specifies a single entity (1) to be revolved into a 3D surface mesh (3). You may choose a line, circle, arc, open or closed polyline, or open or closed spline; 3D objects do not work.

Specifies the entity (2) about which the surface is revolved; you may choose a line or polyline; curved objects do not work.

Specifies the starting angle (4) of the revolution and the angle revolved (5), the number of degrees by which the entity is revolved.







23.42 RIBBON command

Opens the **Ribbon** panel.

Shape Lite Pro Mechanical BIM

23.42.1 Description

Opens the **Ribbon** panel in the current workspace. The **Ribbon** panel appears at the same size and location as it did before it was closed or collapsed. Like any other dockable panel, the **Ribbon** panel can be either floating, docked or stacked.

23.43 RIBBONCLOSE command

Closes the **Ribbon** panel.

Shape Lite Pro Mechanical BIM

23.43.1 Description

Closes the **Ribbon** panel to hide it from the current workspace. If the **Ribbon** panel is stacked when you close it, the Ribbon tab or icon is removed from the stack.

23.44 ROTATE command

Rotates entities about a specified point.

Shape Lite Pro Mechanical BIM

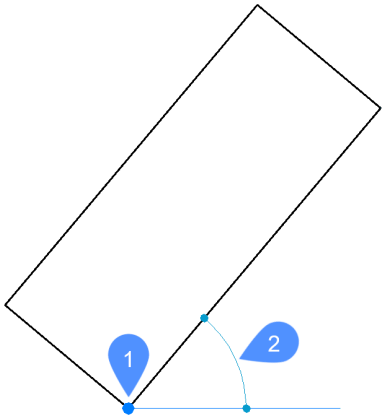
Icon:

Alias: RO

23.44.1 Description

Rotate entities about a specified point at a specified rotation angle, or by an angle referenced to a base angle.

Note: A positive number rotates the entities counter-clockwise. A negative number rotates the entities clockwise. The starting angle is 0 degrees, and the rotation angle is measured from the positive x axis.



1 Rotation point

2 Rotation angle

23.44.2 Options within the command

Base angle

Specifies a new base angle, which is a starting angle other than 0. This option lets you define a different angle as the starting point.

Copy

Rotates a copy of the source entity, instead of rotating the source itself.

23.45 ROTATE3D command

Rotates entities around an axis in 3D space.

Shape Lite Pro Mechanical BIM

Icon:

Alias: 3R

23.45.1 Description

Dynamically rotates 3D solids, surfaces, 2D entities, faces (planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface), edges, or vertices of a solid around an axis.

Note: When the face of a solid or surface is rotated, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.

23.45.2 Method

There are two methods to rotate entities:

- Specify an axis.
- Use the geometry of an object as the axis for rotating the selected entities.
  - Hover over a line or linear polyline segment. Click when the entity highlights.
  - Hover over a 3D solid face, then move the cursor close to the edge you want to use as the rotation axis. A rotation arc and the rotation axis display dynamically. Click to accept the rotation axis.

23.45.3 Options within the command

Xaxis
Rotates the selection around the X-axis of the current coordinate system.

Yaxis
Rotates the selection around the Y-axis of the current coordinate system.

Zaxis
Rotates the selection around the Z-axis of the current coordinate system.

2Points
Defines the rotation axis by two points.

Object
Uses the geometry of an object to determine the axis for rotating the selected entities.
Click to start the rotation of the selected entity. It automatically selects a rotation axis if the cursor is over a planar face.

Last
Uses the last entered axis for rotating the entities.

View
Uses the current view as the axis for rotating the entities.

Note: The axis of rotation is parallel to the direction of view, passing through the picked point.

Rotation angle
Specifies the angle by which to rotate the entities.

Base angle
Allows you to define a new base angle.

Copy
Rotates a copy of the selection set.

Repeat
Creates multiple copies.

Disable connectivity mode
Removes the connectivity with adjacent entities.

Enable connectivity mode
Keeps the connectivity with adjacent entities.

Note: If the HOTKEYASSISTANT system variable is on, press the Ctrl key to cycle between disable/enable connectivity mode.

Note: Geometric and dimensional constraints between selected entities are taken in consideration, as well as the lower and upper bounds parameters for 3D dimensional constraints. Constraints can be accessed and modified through the **Mechanical Browser** panel.

23.46 RSCRIPT command

Reruns the currently loaded SCR script file.

Shape Lite Pro Mechanical BIM

Note: Load and run SCR script files with the Script command.

Note: This command can be entered transparently during commands (**'rscript**).

23.46.1 Method

After a script file is loaded and run, enter RSCRIPT to run it again (short for "repeat script").

23.47 RTEDIT command (Express Tools)

Edits remote text entities.

Shape Lite Pro Mechanical BIM

23.47.1 Options within the command

Style
Specifies the name of the text style to use.

?
Lists the available text styles.

Height
Specifies the height for the RTEXT entity.

Rotation
Specifies a rotation angle for the RTEXT entity.

Edit
Opens a text editor to edit the content of the RTEXT entity.

23.48 RTEXT command (Express Tools)

Inserts or edits a remote text entity.

Shape Lite Pro Mechanical BIM

Icon: 

23.48.1 Description

The RTEXT command can be used to add frequently used text to multiple files. The command inserts the text of a .txt file to the drawing.

23.48.2 Options within the command

Style
Allows you to specify the text style.

?
Allows you to enter the text style mask to list.

*
Lists all available text styles.

Height
Allows you to specify the RText height.

Rotation
Allows you to specify the rotation angle of the RText.

File
Opens the **Select text file** dialog box that allows you to select a .txt file.

Diesel
Opens the **Edit Rtext** dialog box that allows you to use a DIESEL code.

23.49 RTEXTAPP command (Express Tools)

Assigns a specific text file editor used for the RTEXT command.

Shape Lite Pro Mechanical BIM

23.50 RTLOOK command

Looks around in a 3D scene.

Shape Lite Pro Mechanical BIM

23.50.1 Method

The look-around cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to look around in real-time.

23.50.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.51 RTPAN command

Pans the drawing in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.51.1 Method

The pan cursor is displayed during real-time panning. Hold down the mouse's left button and then move the mouse to pan the drawing in real-time.

Note: Hold down the Shift key to restrict panning to the X and Y direction of the viewport.

23.51.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.52 RTROT command

Rotates 3D drawings in a constrained manner, in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.52.1 Method

The constrained rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

Note:

- The ORBITAUTOTARGET system variable determines if the viewpoint rotates around your pick point or the center of the objects.
- The RTISOLATESELECTION system variable controls if the active selection is automatically isolated during real-time rotation.

23.52.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.53 RTROTCTR command

Rotates 3D drawings about a center point that you pick in the drawing, in real-time.

Shape Lite Pro Mechanical BIM

23.53.1 Method

The constrained rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.53.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.54 RTROTF command

Rotates 3D drawings about a fixed point, in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.54.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to look around in real-time.

Note: The ORBITAUTOTARGET system variable determines if the viewpoint rotates around your pick point or the center of the object.

23.54.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.55 RTROTX command

Rotates 3D drawings in real-time about the x axis.

Shape Lite Pro Mechanical BIM

Icon: 

23.55.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

Note: The ORBITAUTOTARGET system variable determines if the viewpoint rotates around your pick point or the center of the object.

23.55.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.56 RTROTY command

Rotates 3D drawings in real-time about the y axis.

Shape Lite Pro Mechanical BIM

Icon: 

23.56.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.56.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.57 RTROTZ command

Rotates 3D drawings in real-time about the z axis.

Shape Lite Pro Mechanical BIM

Icon: 

23.57.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.57.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.58 RTUCS command (Express Tools)

Rotates the UCS dynamically.

Shape Lite Pro Mechanical BIM

23.58.1 Method

Click and drag the screen pointer perpendicular to the active axis to rotate the UCS around it. The UCS is rotated around the highlighted axis at an angle specified by the **Angle** option.

Note: Press the Tab key to change the axis to be rotated. The active axis is dashed. The default active axis is the X-axis.

23.58.2 Options within the command

Save
Saves the current UCS.

Restore
Cycles through the UCSs that were previously saved.

Delete
Deletes the specified UCS from the saved list.

Cycle
Cycles through the standard UCS orientations (Top, Front, Right, Back, Left and Bottom).

Angle
Allows you to specify a minimum rotation increment for the current axis.

Origin
Allows you to set a new origin point for the UCS.

View
Rotates the UCS to align the XY-plane with the current viewing direction. It does not change the UCS origin.

World
Changes the UCS to the World Coordinate System.

Undo
Undoes the UCS changes.

23.59 RTUPDOWN command

Moves the viewpoint up and down, and left and right in a 3D scene, in real time.

Shape Lite Pro Mechanical BIM

23.59.1 Method

The up-down cursor is displayed during real-time movement. Hold down the mouse's left button and then move the mouse to move up-down and left-right in real-time.

23.59.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.60 RTWALK command

Walks left-right and forward-backwards through 3D scenes, in real-time.

Shape Lite Pro Mechanical BIM

23.60.1 Method

The walk cursor is displayed during real-time movement. Hold down the mouse's left button and then move the mouse to walk left-right and forward-backward in real-time.

23.60.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.61 RTZOOM command

Zooms the viewpoint in and out of the drawing in real-time.

Shape Lite Pro Mechanical BIM

Icon:

23.61.1 Method

The zoom cursor is displayed during real-time zooming. Hold down the mouse's left button and then move the mouse to zoom the drawing in real-time.

23.61.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROTF command.

23.62 RULESURF command

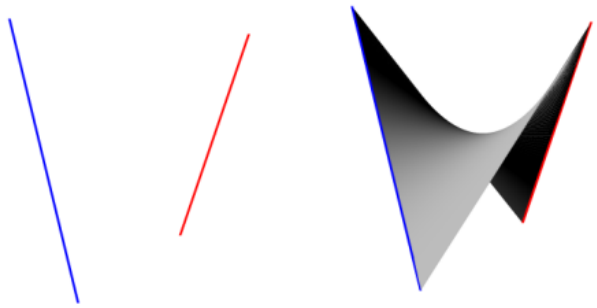
Creates 3D polygon mesh by connecting two separate linear entities (short for "ruled surfaces").

Shape Lite Pro Mechanical BIM

Icon:

23.62.1 Description

Creates a 3D polygon mesh in the shape of a ruled surface by specifying two separate entities.



23.62.2 Method

Specify the first and the second boundary entity. For the first boundary any open object, such as a line, arc, polyline or spline can be selected.

Closed entities and 3D entities are not accepted.

24. S

24.1 SAVE command

Opens the **Save drawing as** dialog box.

Shape Lite Pro Mechanical BIM

Icon:



Alias: SA

24.1.1 Description

Opens the **Save drawing as** dialog box to save data from the current drawing to a DWG, DXF, DWT or DWS file.

File types available to save are:

- **AutoCAD 2018** (*dwg)
- **AutoCAD 2018 ASCII DXF** (*dxf)
- **AutoCAD 2018 Binary DXF** (*dxf)
- **AutoCAD 2013** (*dwg)
- **AutoCAD 2013 ASCII DXF** (*dxf)
- **AutoCAD 2013 Binary DXF** (*dxf)
- **AutoCAD 2010** (*dwg)
- **AutoCAD 2010 ASCII DXF** (*dxf)
- **AutoCAD 2010 Binary DXF** (*dxf)
- **AutoCAD 2007** (*dwg)
- **AutoCAD 2007 ASCII DXF** (*dxf)
- **AutoCAD 2007 Binary DXF** (*dxf)
- **AutoCAD 2004** (*dwg)
- **AutoCAD 2004 ASCII DXF** (*dxf)
- **AutoCAD 2004 Binary DXF** (*dxf)
- **AutoCAD 2000** (*dwg)
- **AutoCAD 2000 ASCII DXF** (*dxf)
- **AutoCAD 2000 Binary DXF** (*dxf)
- **AutoCAD Release 14** (*dwg)
- **AutoCAD Release 14 ASCII DXF** (*dxf)
- **AutoCAD Release 14 Binary DXF** (*dxf)
- **AutoCAD Release 13 Drawing** (*dwg)
- **AutoCAD Release 13 ASCII DXF** (*dxf)

- **AutoCAD Release 13 Binary DXF** (*dxf)
- **AutoCAD Release 11/12 Drawing** (*dwg)
- **AutoCAD Release 11/12 ASCII DXF** (*dxf)
- **AutoCAD Release 11/12 Binary DXF** (*dxf)
- **AutoCAD Release 10 ASCII DXF** (*dxf)
- **AutoCAD Release 10 Binary DXF** (*dxf)
- **AutoCAD Release 9 ASCII DXF** (*dxf)
- **Standards File** (*dws)
- **Drawing Template** (*dwt)

24.2 SAVEALL command

Opens the **Save drawing as** dialog box.

Shape Lite Pro Mechanical BIM

24.2.1 Method

The dialog box is displayed for each open drawing that has been modified since the previous save.

24.3 SAVEAS command

Opens the **Save drawing as** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

24.3.1 Description

Opens the **Save drawing as** dialog box to save data from the current drawing to a DWG, DXF, DWT or DWS file.

Note: File types available for saving are the same as for the SAVE command.

24.4 SAVEASR12 Command

Opens the **Save drawing as** dialog box.

Shape Lite Pro Mechanical BIM

24.4.1 Description

Opens the **Save drawing as** dialog box to save data from the current drawing to an R11/12 DWG file.

24.5 SAVE-CLOSEALL command (Express Tools)

Saves and closes all open drawings.

Shape Lite Pro Mechanical BIM

24.5.1 Method

If changes have been made to any drawings since their last save, the **Save Copy As** dialog box opens to save each drawing before closing them.

Note: For the last open drawing, a BricsCAD dialog box offers you the opportunity to save the drawing before closing it.

24.6 SAVEFILEFOLDER command

Opens the default file explorer.

Shape Lite Pro Mechanical BIM

24.6.1 Description

Opens the default file explorer to quickly access automatic save and backup files. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

24.7 SCALE command

Resizes 2D and 3D entities.

Shape Lite Pro Mechanical BIM

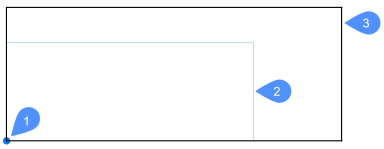
Icon:

Alias: SC

24.7.1 Method

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.

Note: A scale factor smaller than 1 shrinks the entities. A negative scale factor scales the entities in the negative direction.



1 Base point

2 Entity to scale

3 Scaled entity

24.7.2 Options within the command

Reference

Resizes entities using a base or reference scale.

Note: Use this option to scale entities relative to other entities. This command normally scales from a base factor of 1.

Copy
Scales a copy of the selection set, leaving the original entities intact.

24.8 SCALELISTEDIT command

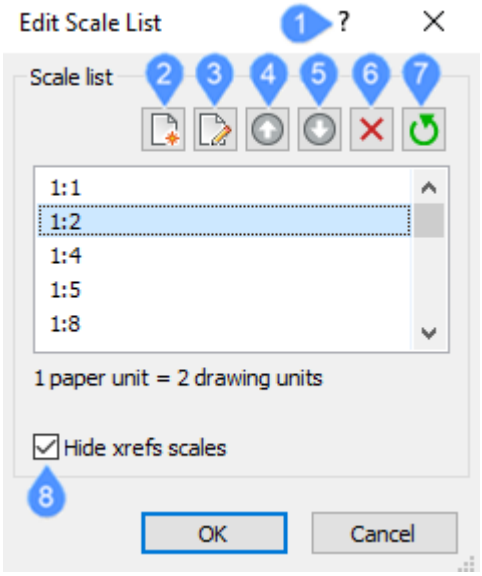
Opens the **Edit scale list** dialog box.

Shape Lite Pro Mechanical BIM

24.8.1 Description

Opens the **Edit scale list** dialog box to edit the list of scales accessible by some commands.

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

24.8.2 Command reference

Opens the Bricsys Help article about the SCALELISTEDIT command.

24.8.3 Add a new scale

Opens the **Add scale** dialog box.

24.8.4 Edit scale

Opens the **Edit Scale** dialog box.

24.8.5 Move up

Moves the selected scale up the list.

24.8.6 Move down

Moves the selected scale down the list.

24.8.7 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.

24.8.8 Reset

Returns the list to its original condition, adding back in the deleted scales and removing all added but unused scales.

24.8.9 Hide xrefs scales

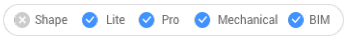
Determines if the scale factors from xrefs are displayed in the list:

- **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.

24.9 -SCALELISTEDIT command

Edits preset scale factors.



24.9.1 Description

Adds and removes scale factors to and from the list used by commands, such as Print, and by annotative scaling; works at the Command line.

24.9.2 Options within the command

? to list
Lists existing preset scale factors in the Prompt History window.

Add
Creates a new scale factor.

Delete
Removes specified scale factor or all unused scale factors.

Reset
Removes all custom scale factors that are not used.

24.10 SCANINTERFERENCES command (Experimental)

Identifies and manages interference between 3D solids or block references in complex 3D models.

Shape Lite Pro Mechanical BIM

24.10.1 Disclaimer

EXP.

The SCANINTERFERENCES command is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

24.10.2 Method

Opens the **Interference Manager** panel if it was previously closed. See also the **Interference Manager panel (Experimental)** article.

24.10.3 Options within the command

select Entities
Defines the selection set.

Select entities in the drawing
Allows you to select entities (3D Solids or Block References) in the drawing.

select Entire drawing
Selects all entities in the drawing.

between Two sets
Searches for interference between two distinct sets of entities.

select First set
Selects the first set of entities for scanning.

select Second set
Selects the second set of entities for scanning.

inside One set
Searches for interference inside a set of entities.

SETtings
Calculates the clash volume.

Yes
Sets value to 1.

No
Sets value to 0.

load Named study
Loads a previously saved scan.

Run Scan
Runs the scanning.

24.11 SCREENSHOT command

Opens the **Save image file** dialog box.

Shape Lite Pro Mechanical BIM

24.11.1 Description

Opens the **Save image file** dialog box to save data from the current drawing to a PNG file. The current view specifies what data to include in the image.

24.12 SCRIPT command

Opens the **Run script** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SCR

24.12.1 Description

Opens the **Run script** dialog box to select an SCR file to run. After choosing **Open** in the dialog box, the script begins running immediately. You can press the **Esc** key to stop it.

24.13 SCROLLBAR command

Toggles the WNDLSCRL system variable.

Shape Lite Pro Mechanical BIM

24.13.1 Description

Toggles the WNDLSCRL system variable to show or hide the drawing window scroll bars. You can launch this command at the Command line or within another command by preceding it with an apostrophe: 'SCROLLBAR.

24.13.2 Options within the command

ON
Turns On the WNDLSCRL system variable.

OFf
Turns Off the WNDLSCRL system variable.

Toggle
Changes the WNDLSCRL system variable to the opposite of the current setting.

24.14 SECTION command

Creates section planes.

Shape Lite Pro Mechanical BIM

Icon:

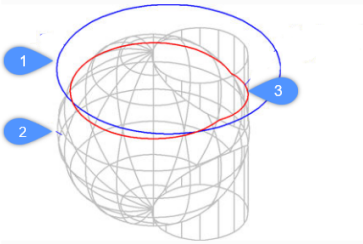
Alias: SEC

24.14.1 Description

Creates section planes of 3D solids, surfaces, polyface meshes, and 3D faces. The result is a slice made of region entities.

24.14.2 Options within the command

Object
Creates the section from entities that intersect the 3D solid.



1 Circle entity

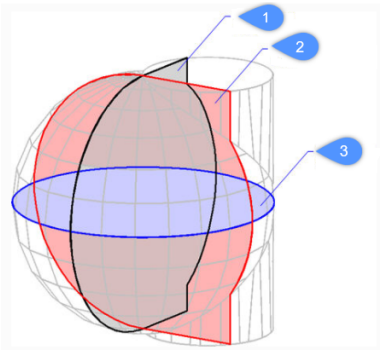
2 3D solid

3 Section defined by the plane of the circle.

Note: The entity defines the plane through which to create the section.

Zaxis
Creates a section in the plane defined by one point on the plane, and a second on the plane's normal (z axis).

View
Creates a section in the plane of the current 3D viewport.



XY (3)
Creates a section parallel to the x, y plane.

YZ (2)
Creates a section parallel to the y, z plane.

ZX (1)
Creates a section parallel to the z, x plane.

3points
Pick points to specify the location of the plane in which the section lies.

Note: Three points define a plane.

24.15 SECTIONPLANE command

Creates section entities.

Shape Lite Pro Mechanical BIM

Icon:

24.15.1 Description

Creates section entities from 3D solids, 3D surfaces, polyface meshes and 3D faces. This command helps you to see inside a 3D entity.

Note: Newly created section planes have their **Clip Display** status set to **On**. As section planes are entities, they can be edited and deleted. Use the SECTIONPLANESETTINGS command to edit the section planes.

24.15.2 Options within the command

Note:

- Use entity snaps to make the pick points accurate.
- Turn **On** ORTHOMODE to create a section plane restricted to the X or Y axis.
- Dynamic dimensions are displayed if **Dynamic Input** (**DYN**) is set to **On** (see **Dynamic dimensions** article).

Specify start point
Specifies the starting point of a 2D section plane.

Note:

- 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.
- BricsCAD ghosts the section plane during the cursor's movement.

select Face
Select the planar face of a 3D solid.

Note:

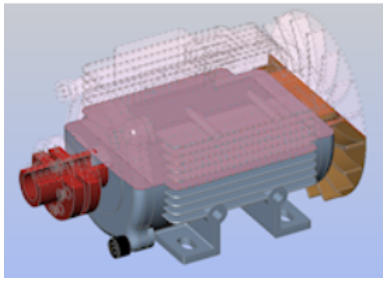
- The section plane coincides with the selected face. The **Live Section** property turns **On** automatically. Live sections are sections that can be interactively edited through the **Properties** panel.
- BricsCAD ghosts the section plane during the cursor's movement.

Draw
Draws polygonal-shaped section planes. Define the section plane by two or more points.

Note: The section plane is created through the specified points and perpendicular to the current **UCS**.

Orthographic
Places a section plane entity in one of the six standard drawing orientations, relative to the current **UCS**.

Note: BricsCAD draws the orthographic section plane through the center of the bounding box that contains all 3D entities in the drawing. For example, the result of a **Top** section plane:



24.16 SECTIONPLANESETTINGS command

Shape Lite Pro Mechanical BIM

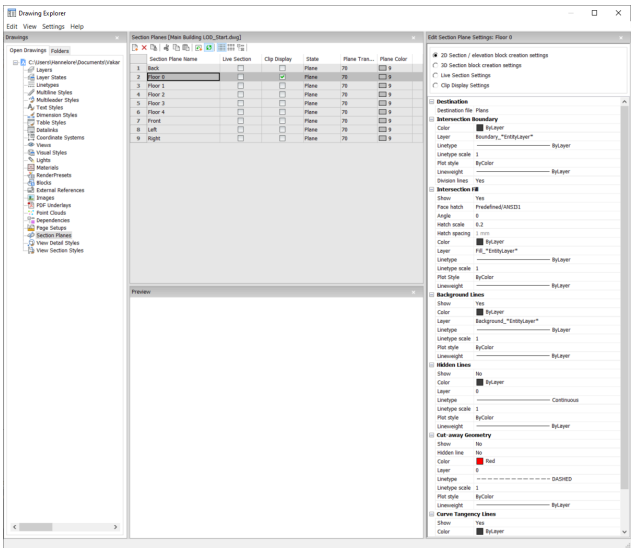
Icon:

24.16.1 Description

Opens the **Drawing Explorer** dialog box with Section Planes category selected.

24.16.2 Methods

Open the **Drawing explorer** dialog box with Section Planes category selected to view and modify section planes in the current drawing.



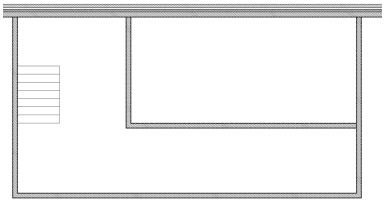
Note: The available options differ depending on the kind of section you choose to edit.

24.16.3 Options within the command

Section Types

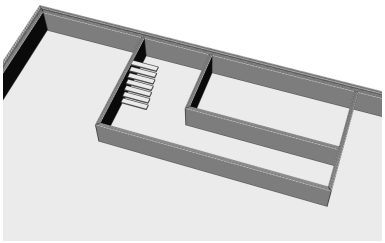
2D Section/elevation block creation settings

Specifies properties for 2D sections and elevations. These properties are assigned to the 2D section you create as a block.



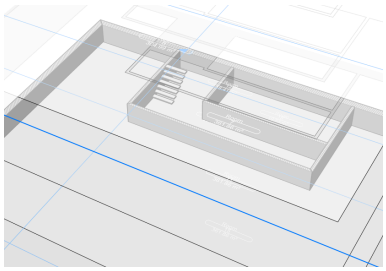
3D section block creation settings

Specifies properties for 3D sections. These properties are assigned to the 3D section you create as a block.



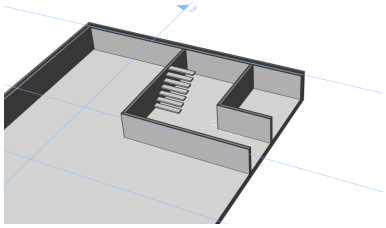
Live Section Settings

Specifies properties for live sections. With live sections, the entities in the drawing are temporarily modified and viewable in the drawing.



Clip Display Settings

When using the clip display status, the display of entities is clipped. The status can be set for multiple section entities simultaneously.



Destination

Destination file

Specifies the destination file to save the section plane to.

Intersection Boundary

Color

Defines the color of the intersection boundary. You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer

Specifies the layer of the intersection boundary. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above.

Linetype

Specifies the linetype of the intersection boundary. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale

Specifies the linetype scale of the intersection boundary.

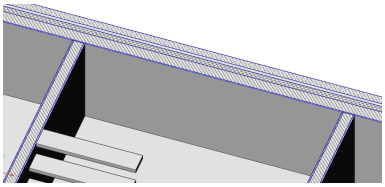
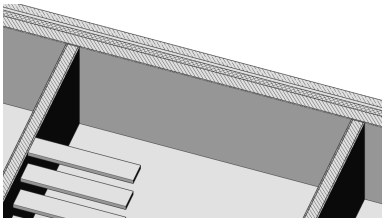
Plot style

Specifies the plotstyle of the intersection boundary.

Lineweight
Specifies the lineweight of the intersection boundary.

Division lines (option available for the 2D sections)
Specifies whether the division lines of the intersection boundary should be drawn.

Show (option available for the 3D sections)
Specifies whether the intersection boundary is drawn.

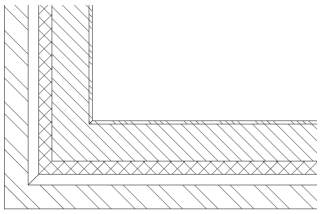
- Yes: the intersection boundary is displayed. The intersection boundaries are indicated in blue.
- 
- No: no intersection boundary is displayed.
- 

  Note: The intersection boundary is still visible, since the lines are also drawn for the intersection fill. The color is now determined by the intersection fill instead of by the intersection boundary.

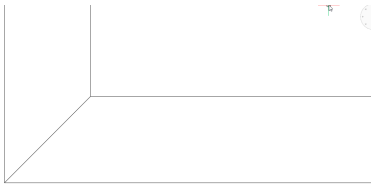
Intersection Fill

Show
Specifies whether the intersection filling is showed.

- Yes: the cut through geometry is displayed by a hatch



- No: no hatch is drawn on the cut through geometry



Face hatch
Specifies the used hatch pattern to fill the section. Opens the **Hatch Pattern Type** dialog box where you can choose a hatch pattern.

Angle
Specifies the angle of the hatch pattern.

Hatch scale
Specifies the scale of the hatch pattern.

Hatch spacing
Specifies the hatch spacing.

Color
Specifies the color of the hatch. You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer
Specifies the layer of the intersection boundary. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the intersection boundary. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the hatch pattern.

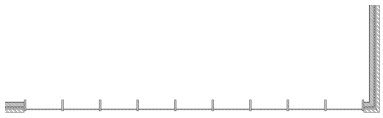
Plot Style
Specifies the plot style of the hatch pattern.

Lineweight
Specifies the lineweight of the hatch pattern.

Background Lines

Show
Specifies whether the background lines are drawn. These are the lines that show the geometry behind the section plane, which is not cut-through.

- Yes: the background lines are displayed
- 
- No: no background lines are displayed

- 

Color
Specifies the color of the background lines. You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer
Specifies the layer of the background lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the background lines. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the background lines.

Plot Style
Specifies the plot style of the background lines.

Lineweight
Specifies the lineweight of the background lines.

Hidden lines

Show
Specifies whether the hidden lines are drawn.

- Yes: the hidden lines are displayed.
- 
- No: no hidden lines are displayed.
- 

Color
Specifies the color of the hidden lines. You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer
Specifies the layer of the hidden lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the hidden lines. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the hidden lines.

Plot Style
Specifies the plot style of the hidden lines.

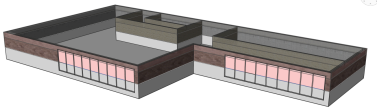
Lineweight
Specifies the lineweight of the hidden lines.

Cut-away Geometry

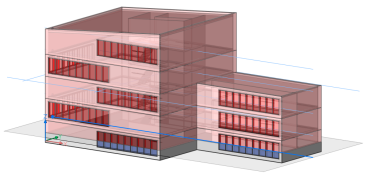
Show
Specifies whether the section lines of the geometry that is cut away are shown.

- Yes: the cut-away geometry is displayed.
- 

  Cut-away geometry of a 2D section.

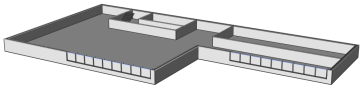
  Cut-away geometry of a 3D section.

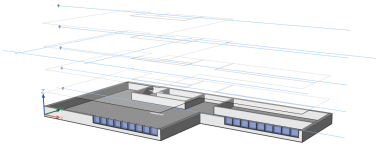
  Cut-away geometry of a live section.
- No: no cut-away geometry lines are displayed

- 

  No cut-away geometry of a 2D section is shown.

  No cut-away geometry of a 3D section is shown.

  No cut-away geometry of a live section is shown.

Hidden line
Specifies whether the hidden lines of the cut-away geometry are shown.

Color
Specifies the color of the cut-away geometry (lines). You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer
Specifies the layer of the cut-away geometry. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the cut-away geometry. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the cut-away geometry lines.

Plot Style
Specifies the plot style of the cut-away geometry lines.

Lineweight
Specifies the lineweight of the cut-away geometry lines.

Curve Tangency Lines

Show
Specifies whether the curve tangency lines are drawn.

Color
Specifies the color of the curve tangency lines. You can choose a color from the drop list or choose Select Color, the **Color** dialog box will then display.

Layer
Specifies the layer of the curve tangency lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the **New Layer Name** dialog box. Here you can edit the name settings for the option above. See the related article **New Layer Name dialog box**.

Linetype
Specifies the linetype of the curve tangency lines. You can choose between:

- The available linetypes in the drawing
- Load new linetypes. This opens the **Load Linetypes** dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the curve tangency lines.

Plot Style
Specifies the plot style of the curve tangency lines.

Lineweight
Specifies the lineweight of the curve tangency lines.

24.16.4 Context Menu Options

New
Creates a new section plane definitions into the drawing. For a detailed explanation, see the SECTION-PLANE command.

Delete
Deletes section planes from the drawing.

Rename
Renames the selected section plane.

Select All
Selects all section plane definitions.

Invert selection
Deselects the current selection and vice versa.

24.17 SECTIONPLANETOBLOCK command

Opens the **Create section/elevation** dialog box.

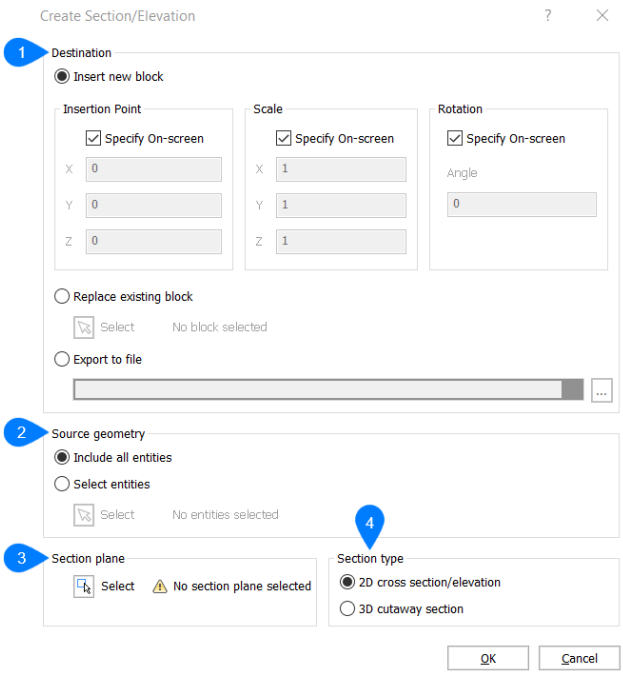
Shape Lite Pro Mechanical BIM

Icon:

24.17.1 Description

Opens the **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

24.17.2 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.

24.17.3 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).

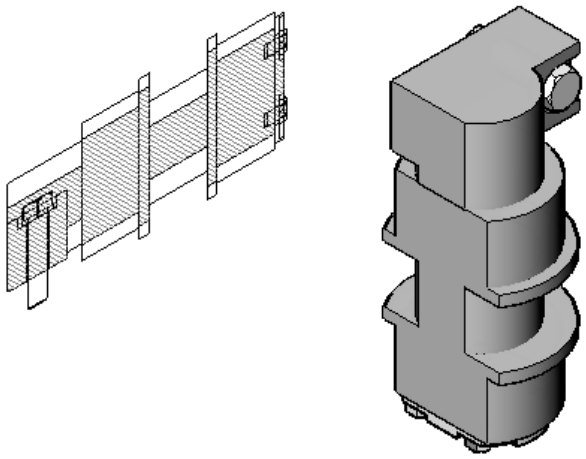
Select entities
Allows to select the entities to be included in the model space.

24.17.4 Section plane

Selects the section plane to be converted to a block.

24.17.5 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.

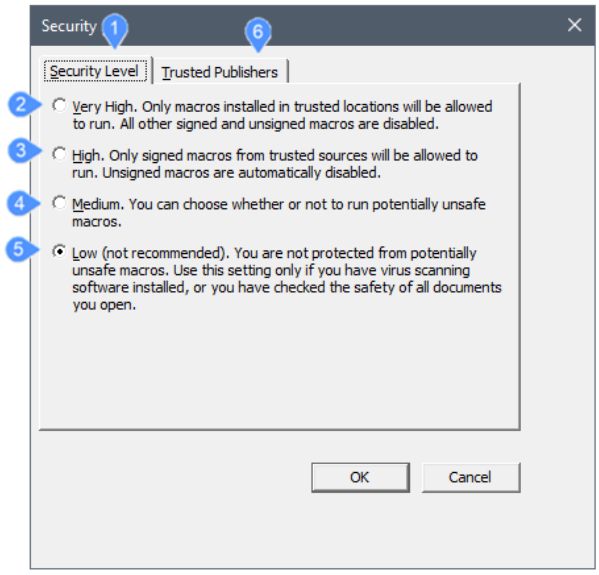
24.18 SECURITY command

Opens the **Security** dialog box.

Shape Lite Pro Mechanical BIM

24.18.1 Description

Opens the **Security** dialog box to specify if VBA macros are allowed to run automatically by selecting a security level.



1 Security Level

2 Very High

3 High

4 Medium

5 Low

6 Trusted Publishers

24.18.2 Very High

Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.

24.18.3 High

Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.

24.18.4 Medium

BricsCAD displays a dialog box when a VBA macro runs.

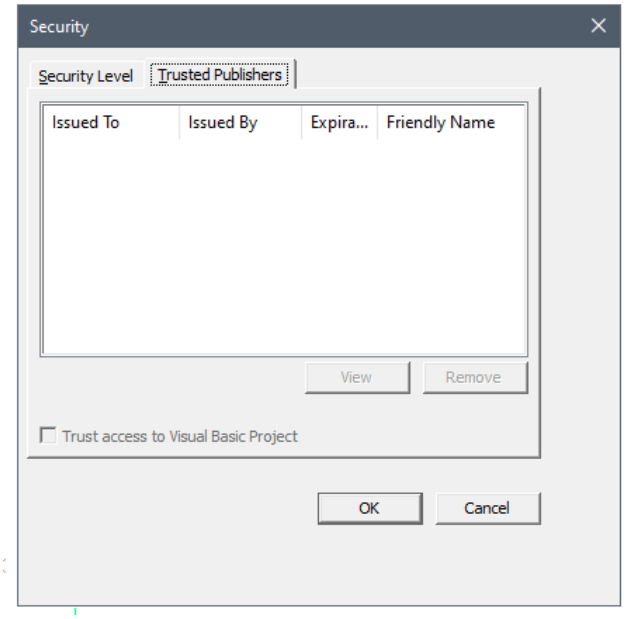
The dialog box allows you to choose whether or not to run the macro.

24.18.5 Low

All VBA macros are run.

24.18.6 Trusted Publishers

Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High.



Remove
Removes the selected VBA macro from the list.

Trust access to Visual Basic Project
Allows access to the VBA Project.

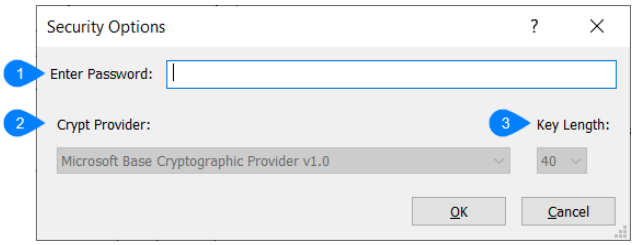
24.19 SECURITYOPTIONS command

Opens the **Security options** dialog box.

Shape Lite Pro Mechanical BIM

24.19.1 Description

Opens the **Security options** dialog box to specify a password to protect your drawing from unauthorized access.



1 Enter Password

2 Crypt Provider

3 Key Length

24.19.2 Enter Password

Specify a password by entering one or more characters.

24.19.3 Crypt Provider

Specifies the encryption level for drawing properties. Choose an encryption provider supplied by the operating system.

24.19.4 Key Length

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

24.20 SELECT command

Selects entities.

Shape Lite Pro Mechanical BIM

Icon:

24.20.1 Description

Selects one or more entities through a variety of selection methods.

Note: You access the selection set during subsequent commands with the Previous option.

24.20.2 Methods

There are two methods to select entities:

- Select entities separately using the command.
- Crtl+A selects all entities on non-frozen layers.

24.20.3 Options within the command

Select entities to include in set
Chooses entities to add to the selection set; use one of methods listed below. As you add entities to the selection set, the program highlights them with dashes.

select ALL entities
Selects all entities in the drawing, except for those on frozen layers.

Add
Adds additional entities to the selection set. (Used after the Remove option by typing Add in the Command line.)

add to set (+)
Adds additional entities to the selection set. (Used after the Remove option by typing + in the Command line.)

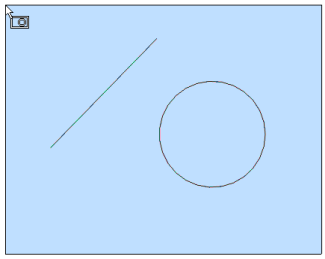
Remove
Removes entities from the selection set. As you remove entities from the selection set, the program unhighlights them.

subtract from set (-)
Removes entities from the selection set. As you remove entities from the selection set, the program unhighlights them.

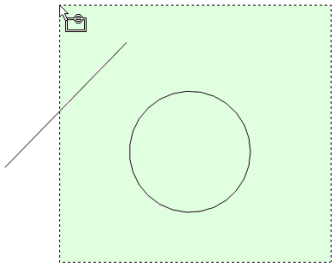
Previous selection
Add the entities from the previous selection to the current select set.

Last entity in drawing
Adds the last entity drawn to the selection set.

inside Window
Selects all entities that are entirely inside a rectangular selection window. The selection window outline is drawn, filled with transparent blue.

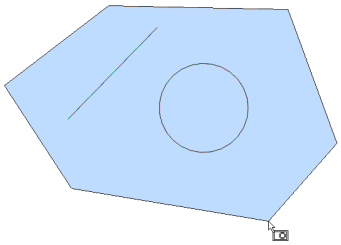


Crossing window
Selects all entities that are inside of and cross a rectangular selection window. The selection window outline is drawn in a dashed line, filled with transparent blue.

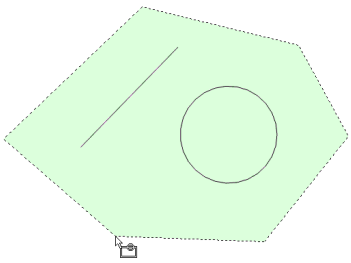


Outside window
Selects all entities fully outside a rectangular selection window.

Window Polygon
Selects all entities fully inside a polygonal selection window. The selection polygon outline is drawn, filled with transparent blue.

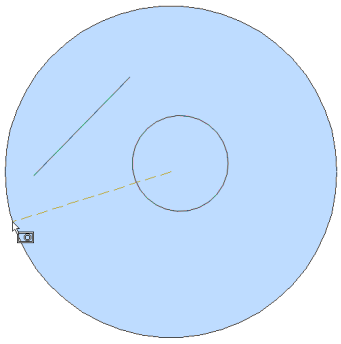


Crossing Polygon
Selects all entities inside and crossing a polygonal selection window. The selection polygon outline is drawn with a dashed line, filled with transparent green.



Outside Polygon
Selects all entities fully outside a polygonal selection window.

Window Circle
Selects all entities fully inside a circular selection window. The selection circle outline is drawn, filled with transparent blue.



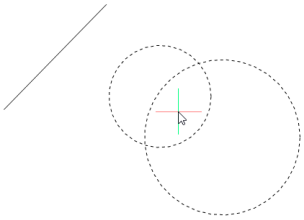
Crossing Circle
Selects all entities inside and crossing a circular selection window. The selection circle outline is drawn with a dashed line, filled with transparent green.

Outside Circle
Selects all entities fully outside a circular selection window.

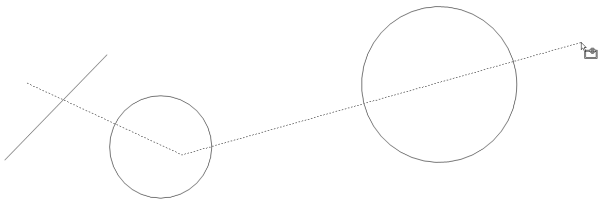
Box
Selects entities depending on the direction you move the cursor:

- Right to left - Box acts like Crossing mode.
- Left to right - Box acts like Window mode.

POint
Selects all closed entities surrounding the pick point, such as all walls of a room.



Fence
Selects all entities that cross the fence selection lines.



AUto
Selects entities depending on how you use the cursor:

- Pick an entity - the entity is selected
- Pick two points right to left - Auto acts like Crossing mode.
- Pick two points left to right - Auto acts like Window mode.

Multiple
All entities selected during this mode are added to the selection set.

Single
After you select a single entity, the command ends.

select by PROperties…
Selects entities based on their properties.

Note: For the panel version of this option, see the QSELECT command.

Color
Selects entities based on their color.

Note: :If there are entities colored in a specific color because the color properties of the layer or the block are on that specific color, these entities will not be selected.

COlorbook
Selects entities with a colorbook color.

Note: If there are entities colored in a colorbook color because the color properties of the layer or the block, these entities will not be selected.

Layer
Selects entities based on their layer. Enter * to select all layers.

LineType
Selects entities based on their linetype. Enter * to select all linetypes.

Name
Selects entities based on their entity name. Enter * to select all entity names.

Thickness
Selects entities based on their thickness.

Type
Selects entities based on their entity type, such as line, circle, arc... Enter * to select all entity types.

Value
Selects entities based on their value. Enter * to select all values.

Width
Selects entities based on the width of the entities.

Handle
Selects entities based on the value of a handle. This is a hexadecimal number that uniquely identifies each entity in the drawings; the List command reports handle numbers.

Location
Returns to the 'Select entities to include in set' prompt.

selection methoDs…
Displays the Entity Selection section of the **Settings** dialog box.

Undo
Undoes the last action during this command.

Group
Adds a group to the selection set. See the GROUP command.

Note: In macros, the ^S metacharacter selects the entity under the cursor automatically.

24.21 SELECTALIGNEDFACES command

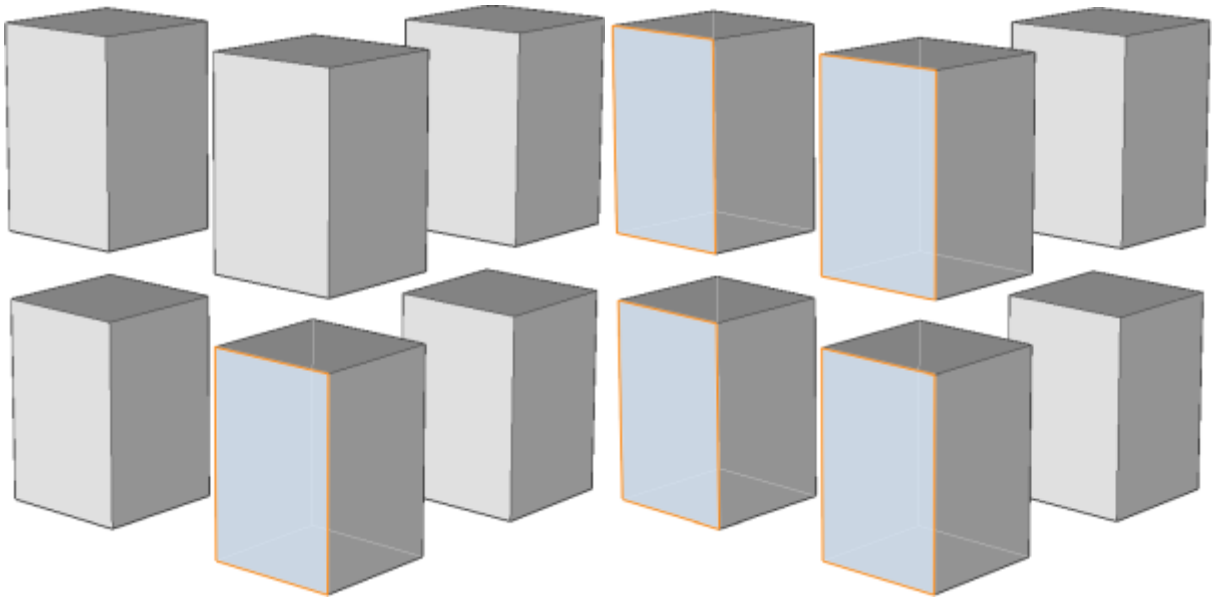
Selects all faces in a model that are coplanar with a selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.21.1 Method

Select a single face. The program selects all other faces that are in the same plane, shown outlined.



24.22 SELECTALIGNEDSOLIDS command

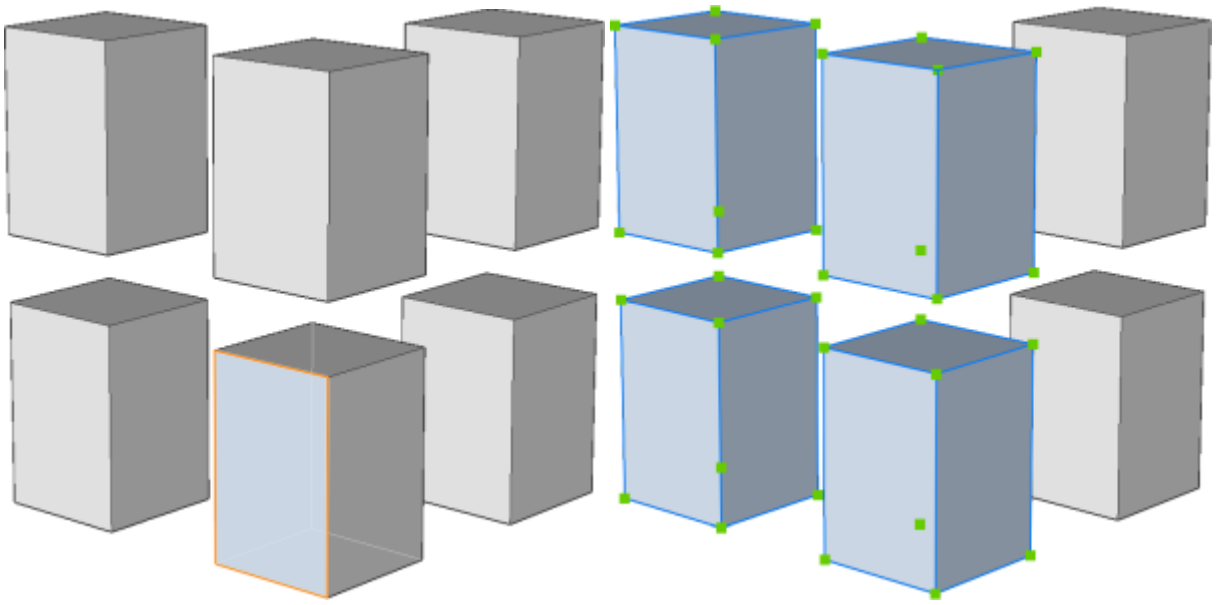
Selects all solids in a model that have faces coplanar with the selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.22.1 Method

Select a single face. The program selects all solids that have faces in the same plane, shown outlined.



24.23 SELECTCONNECTEDFACES command

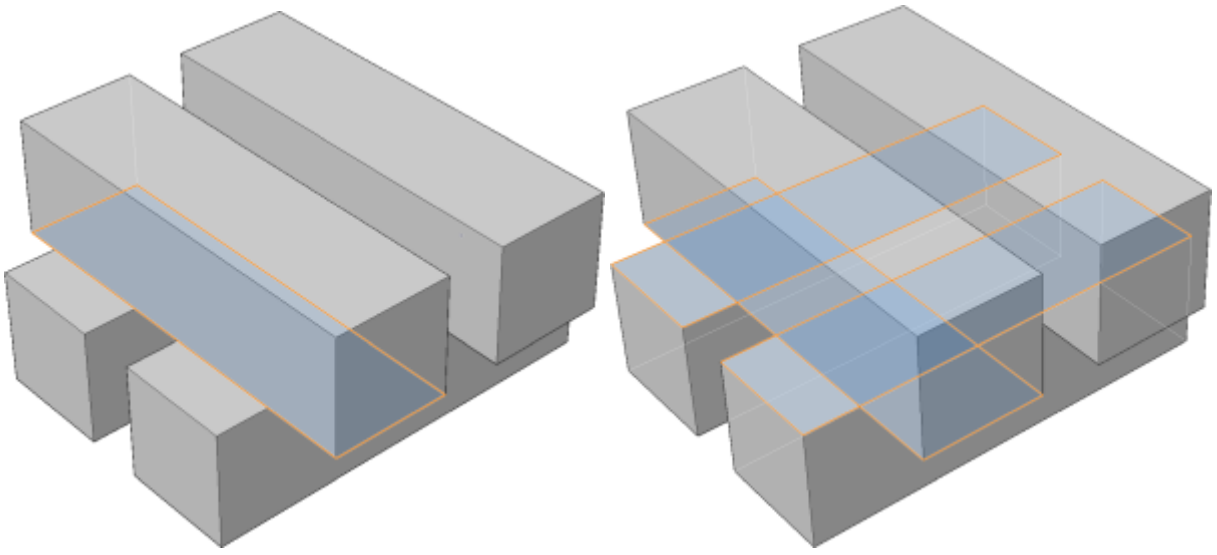
Selects all faces in a model that are connected to a selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.23.1 Method

Select a face. The program selects all other faces that are connected to the selected face, shown outlined.



24.24 SELECTCONNECTEDSOLIDS command

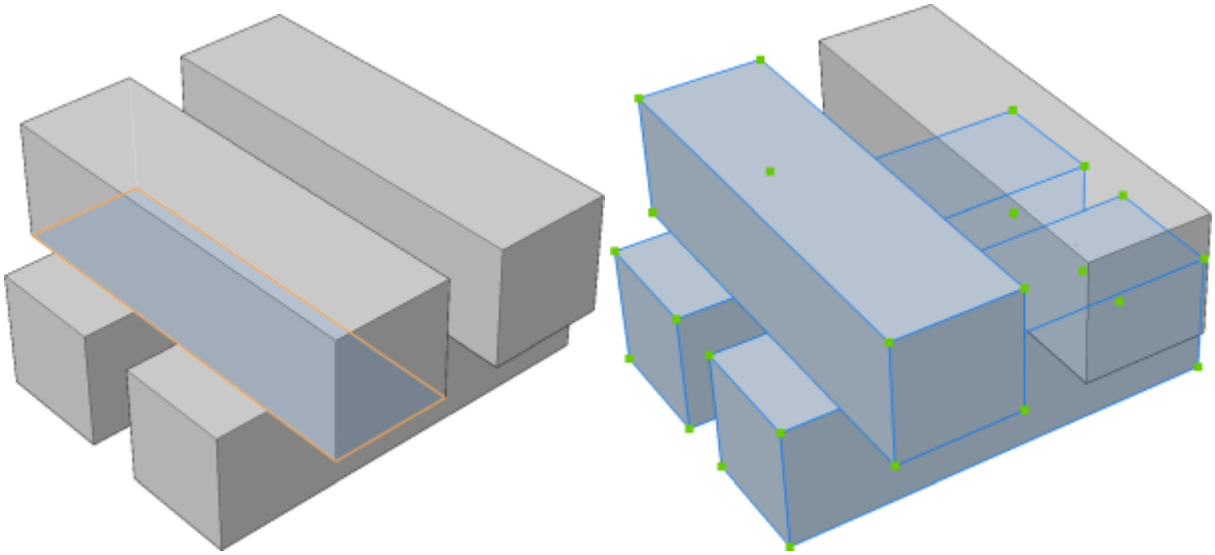
Selects all solids in a model that are connected to a selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.24.1 Method

Select a face. The program selects all solids that are connected to the selected face, shown outlined.



24.25 SELECTSIMILAR command

Selects similar entities.

Shape Lite Pro Mechanical BIM

Icon:

24.25.1 Description

Selects all entities of the same type (lines, polylines, circles, etc.) as the selected ones, having the same properties as those defined in the **SEttings** option dialog box.

24.25.2 Method

Select one or more entities and choose the properties based on which the similar entities will be selected. The properties selected by default are **Layer** and **Name**. Use the **SEttings** option to choose properties. These settings are stored by the SELECTSIMILARMODE system variable.

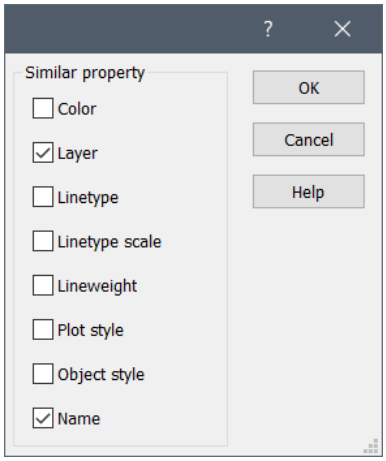
When you exit the command, the entities that match these types and properties are added to the selection set (highlighted and their entity grips displayed).

Note:

- For this command to operate as intended, at least one property must be selected.
- You can edit the selection set further by using the QSELECT command.

24.25.3 Options within the command

SEttings
Displays a dialog box to select entity properties.



Note: The value of the SELECTSIMILARMODE system variable will be updated according to the selected properties in this dialog box.

Color
Adds the entities that have similar colors to the selection set.

Layer
Adds the entities on matching layers to the selection set.

Linetype
Adds the entities with matching linetype to the selection set.

Linetypescale
Adds the entities with matching linetypescale to the selection set.

Plot style
Adds the entities with matching plot style to the selection set.

Object style
Adds the entities with matching styles (such as text styles, dimension styles, and table styles) to the selection set.

Name
Adds referenced entities (such as blocks, xrefs, and images) with matching names to the selection set. Also, adds unnamed entities of the same type (such as lines and circles) to the selection set.

24.26 SELGRIPS command

Displays entities grips.

Shape Lite Pro Mechanical BIM

24.26.1 Description

Selects entities and then displays their grips.

BricsCAD highlights selected entities and displays their grips.

Edit the entities by dragging the grips. To select all entities in the drawing, press **Ctrl+A**.

24.27 SETBYLAYER command

Resets overridden properties to BYLAYER.

Shape Lite Pro Mechanical BIM

24.27.1 Description

Changes the value of overridden properties (color, linetype, lineweight, material, plot style, and transparency) back to the default value of BYLAYER.

24.27.2 Method

Select the entities whose properties will be set as BYLAYER according to the value of the SETBYLAYERMODE system variable.

Enter 'all' to select all non-frozen entities in the current viewport.

When entities contain blocks the BricsCAD asks whether blocks should have their properties changed.

24.27.3 Options within the command

Settings

Allows to select the entity properties to be changed from the SetByLayer settings dialog box:

- Color - resets the color of the selected entities to BYLAYER.
- Linetype - resets the linetype of the selected entities to BYLAYER.
- Lineweight - resets the lineweight of the selected entities to BYLAYER.
- Material - resets the material of the selected entities to BYLAYER.
- Plot Style - (only in drawings with plot styles) resets the plot style of the selected entities to BYLAYER.
- Transparency - resets the transparency value of the selected entities to BYLAYER.

Note: The settings are saved through the SETBYLAYERMODE system variable.

Note: The Plotstyle option is available for STB drawings only. See the CONVERTCTB and CONVERTPSTYLES command.

24.28 -SETBYLAYER command

Sets the property overrides of selected entities to ByLayer.

Shape Lite Pro Mechanical BIM

24.28.1 Description

Changes the value of overridden properties (color, linetype, lineweight, material, plot style, and transparency) back to the default value of BYLAYER.

24.28.2 Method

Select the entities whose properties will be set as BYLAYER according to the value of the SETBYLAYERMODE system variable.

Enter 'all' to select all non-frozen entities in the current viewport.

When entities contain blocks the BricsCAD asks whether blocks should have their properties changed.

24.28.3 Options within the command

Settings

Allows to select the entity properties to be changed from the command line:

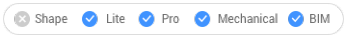
- **Color**: resets the color of the selected entities to BYLAYER.
- **Linetype**: resets the linetype of the selected entities to BYLAYER.
- **Lineweight**: resets the lineweight of the selected entities to BYLAYER.
- **Material**: resets the material of the selected entities to BYLAYER.
- **Plot Style**: (only in drawings with plot styles) resets the plot style of the selected entities to BYLAYER.
- **Transparency**: resets the transparency value of the selected entities to BYLAYER.

Note: The settings are saved through the SETBYLAYERMODE system variable.

Note: The Plotstyle option is available for STB drawings only. See the CONVERTCTB and CONVERTPSTYLES command.

24.29 SETLICENSELEVEL command

Changes the available commands by setting the license type.



24.29.1 Description

Runs BricsCAD and/or Communicator for BricsCAD® at different license levels, which affects the commands that are available to the user. For the new license level to take effect, users must exit BricsCAD and then start it again.

24.29.2 Method

There are two methods to change the license level:

- Change the license level of BricsCAD.
- Change the license level of Communicator for BricsCAD®.

24.29.3 Options within the command

Run BricsCAD at another license level

- **Lite** - runs BricsCAD without 3D functionalities, except API and LISP.
- **Pro** - runs BricsCAD without BIM and Mechanical commands.
- **Bim** - runs BricsCAD with Pro and BIM commands.
- **Mechanical** - runs BricsCAD with Pro and Mechanical commands.

- **Ultimate** - runs BricsCAD with full functionalities, including BIM and Mechanical commands.

Run Communicator at another license level

- **No license** - disables Communicator for BricsCAD®.
- **Trial** - stops running Communicator for BricsCAD® after 30 days.
- **Full** - runs Communicator for BricsCAD® normally.

24.30 SETPROJECTLOCATION command

Adds a marker for the project location.

Shape Lite Pro Mechanical BIM

24.30.1 Description

Sets the Project location point coordinates and angle.

Note: The Project location points coordinates and angle will be used in RVT and IFC import and export.

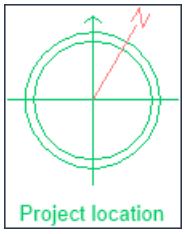
24.30.2 Method

Add a Project location point coordinates and angle to a new drawing or edit the Project location point information in an existing drawing.

Pick a point in the drawing or enter the point coordinates manually and specify an angle.

Note: The default values of the point coordinates and angle in a new drawing are 0. The current values are displayed in the Command line when running the command.

Toggle On/Off the visibility of the Project location marker by modifying the PROJECTLOCATIONVISIBILITY system variable's value (On by default).



Note: The True North indicator displays the NORTHDIRECTION angle measured clockwise, stored by the NORTHDIRECTION system variable, and is the same as the Survey location angle.

24.30.3 Options within the command

Point
Sets the X, Y, and Z coordinates of the point for project location marker.

Angle
Sets the angle between Project Y-axis and Drawing Y-axis, clockwise.

Clear
Clears the previously set point coordinates and angle (sets these values back to 0).

24.31 SETSITELOCATION command

Adds a marker for the site location.

Shape Lite Pro Mechanical BIM

24.31.1 Description

Sets the Site location point coordinates and angle.

Note: The site location point coordinates and angle will be used in RVT and IFC import and export.

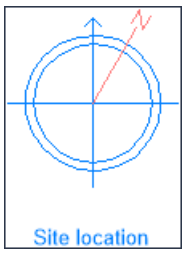
24.31.2 Method

Add a Site location point coordinates and angle to a new drawing or edit the Site location point information in an existing drawing.

Pick a point in the drawing or enter the point coordinates manually and specify an angle.

Note: The default values of the point coordinates and angle in a new drawing are 0. The current values are displayed in the Command line when running the command.

Toggle On/Off the visibility of the Site location marker by modifying the SITELOCATIONVISIBILITY system variable's value (On by default).



Note: The True North indicator displays the NORTHDIRECTION angle measured clockwise, stored by the NORTHDIRECTION system variable, and is the same as the Survey location angle.

24.31.3 Options within the command

Point
Sets the X, Y, and Z coordinates of the point for site location marker.

Angle
Sets the angle between Site Y-axis and Drawing Y-axis, clockwise.

Clear
Clears the previously set point coordinates and angle (sets these values back to 0).

24.32 SETSURVEYLOCATION command

Adds a marker for the survey location.

Shape Lite Pro Mechanical BIM

24.32.1 Description

Sets the Survey location point coordinates and angle.

Note: The Survey location point coordinates and angle will be used in RVT and IFC import and export.

24.32.2 Method

Add a Survey location point coordinates and angle to a new drawing or edit the Survey location point information in an existing drawing.

Pick a point in the drawing or enter the point coordinates manually and specify an angle.

Note: The default values of the point coordinates and angle in a new drawing are 0. The current values are displayed in the Command line when running the command.

Toggle On/Off the visibility of the Survey location marker by modifying the GEOMARKERVISIBILITY system variable's value (On by default).



Note: The True North indicator displays the NORTHDIRECTION angle measured clockwise, stored by the NORTHDIRECTION system variable, and is the same as the Survey location angle.

Note:

- The Survey Location point corresponds to the Geographic Location point. You can find its coordinates in the **Position** section of the **Geographic Location** dialog box (see the **GEOGRAPHICLOCATION command** article).
- The angle value set for the Survey Location point corresponds to the NORTHDIRECTION system variable's value and represents the angle between the True North and the Y-axis, measured clockwise. Modifying the NORTHDIRECTION system variable's value changes the Survey Location angle value only if a **GIS Coordinate System** is set in the **Geographic Location** dialog box.

24.32.3 Options within the command

Point
Sets the X, Y, and Z coordinates of the point for survey location marker.

Angle
Sets the angle between True North and Drawing Y-axis, clockwise.

Clear
Clears the previously set point coordinates and angle (sets these values back to 0).

24.33 SETTINGS command

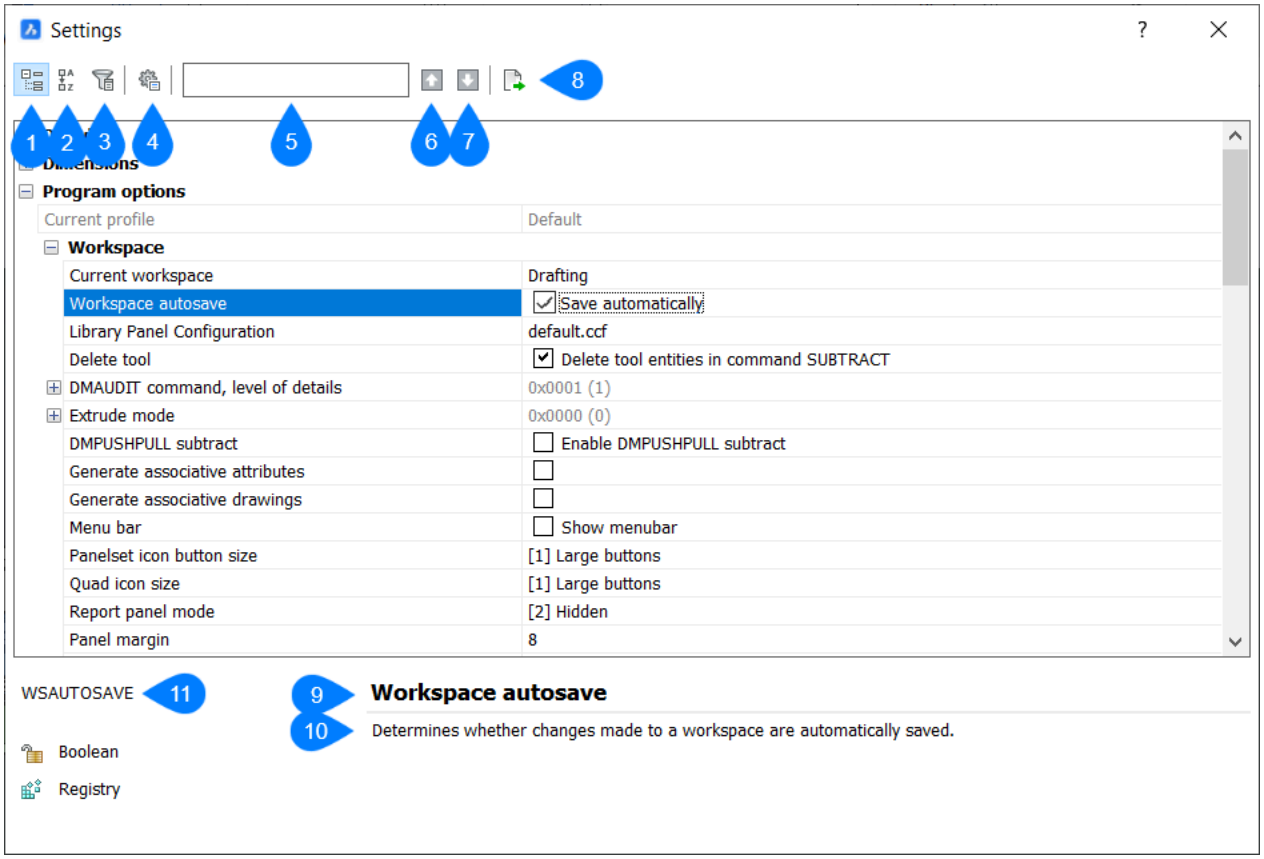
Opens the **Settings** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

24.33.1 Description

Opens the **Settings** dialog box to view and modify system variables. Most, but not all, system variables are available in the **Settings** dialog box. You can modify all system variables using the SETVAR command.



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

24.33.2 Categorized

Lists variables in related groups.

24.33.3 Alphabetic

Lists the variables alphabetically.

24.33.4 Show differences

Lists variables whose values are changed from the initial (default) values.

24.33.5 Dialog configurations

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.

24.33.6 Search bar

Lists variables through a real-time search field when entering the first few letters of their names or descriptions.

24.33.7 Next

Jumps to the next candidate that matches the search phrase.

24.33.8 Previous

Jumps to the previous candidate that matches the search phrase.

24.33.9 Export

Opens the **Export Settings** dialog box to allow you to save all settings and values to a CSV file.

24.33.10 Name

Displays the name of the selected system variable.

24.33.11 Description

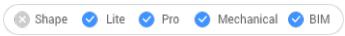
Displays a brief description of the selected system variable.

24.33.12 Properties

Lists several basic properties/characteristics of the selected system variable.

24.34 SETTINGSSEARCH command

Opens the **Settings** dialog box only for a list of words specified by the user.



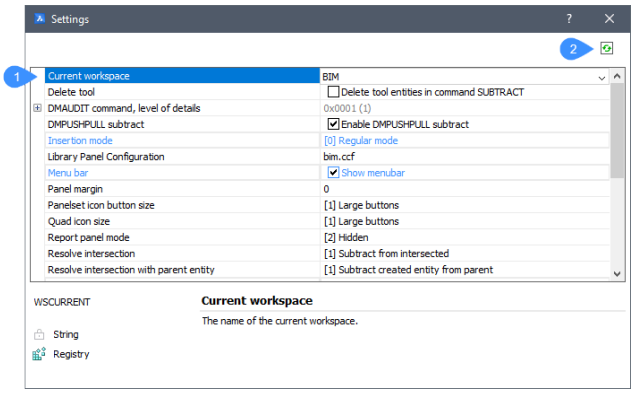
24.34.1 Description

Searches the **Settings** dialog box for a user-specified list of words separated by commas or semicolons, and displays only those filtered variables.

24.34.2 Method

1 Enter a list of words separated by commas or semicolons. The search words will be matched to the following items:
   - Category titles
   - Variable titles, names, and values
   - Variable help text

2 The **Settings** dialog box will only display system variables that contain the specified words:



Search result (1)
Displays the filtered system variables which contain at least one of the specified word in the list.

Toggle button (2)
Toggles the display of **filtered variables** to **all variables** in the **Settings** dialog box.

24.35 SETUCS command

Opens the **User Coordinate System** dialog box.

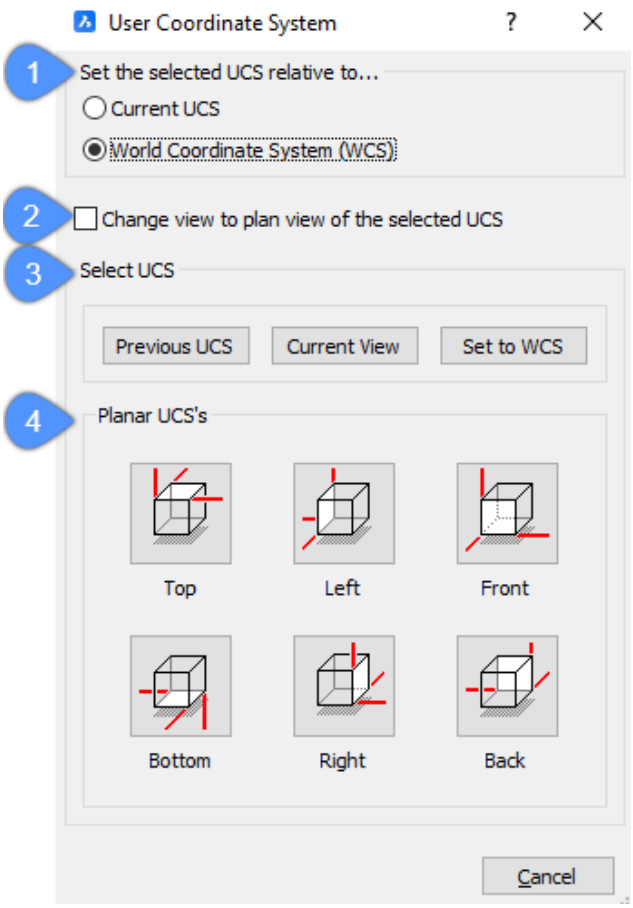
Shape Lite Pro Mechanical BIM

Icon:

Alias: DDUCSP, UCP

24.35.1 Description

Opens the **User Coordinate System** dialog box to specify a viewpoint to use for the current UCS (User Coordinate System).



1 Set the selected UCS relative to...

2 Change view to plan view of the selected UCS

3 Select UCS

4 Planar UCS's

24.35.2 Set the selected UCS relative to...

Specifies how the new UCS is set:

- **Current UCS**: Sets the new UCS relative to the current UCS.
- **World Coordinate System**: Sets the new UCS relative to the WCS. World Coordinate System is the default coordinate system when no UCS is set and it cannot be changed.

24.35.3 Change view to plan view of the selected UCS

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.

24.35.4 Select UCS

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 WCS**: Changes the UCS to match the WCS.

24.35.5 Planar UCS's

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.

24.36 SETVAR command

Displays and changes the values of system variables.

Shape Lite Pro Mechanical BIM

Alias: SET

24.36.1 Description

Displays and changes the values of system variables at the Command line.

Note: This command can be entered transparently: 'SETVAR.

Note: The names of system variables can also be entered directly at the Command line, without using this command.

24.36.2 Method

There are two methods to use SETVAR command:

- Enter the name of a system variable to see its state.
- Choose to display a list that contains all the system variable.

24.36.3 Options within the command

Variable name

Specify the name of a system variable.

?

Lists the names of system variables.

*
Lists all system variables.

Name*
Lists all system variables starting with Name.

24.37 SHADE command

Generates shaded images of 3D drawings.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SHA

24.37.1 Description

Generates shaded images of the current 3D drawing, based on the current setting of the SHADEMODE command. Shaded images are simpler versions of visual styles and rendered images.

24.38 SHADEMODE command

Specifies the shading style for the current drawing.

Shape Lite Pro Mechanical BIM

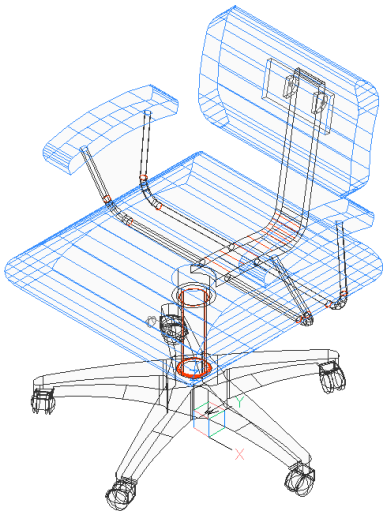
24.38.1 Description

Specifies the shading style for the current drawing to be used by the SHADE command.

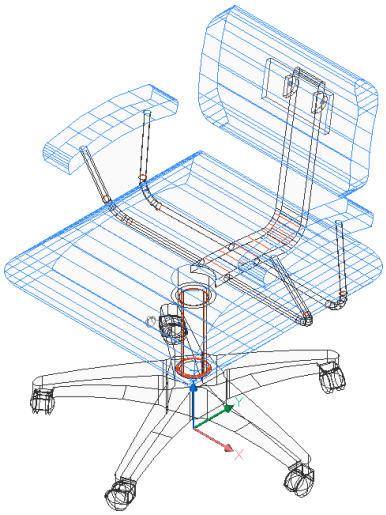
24.38.2 Options within the command

Enter visual style
Allows you to choose the name of a preset visual style.

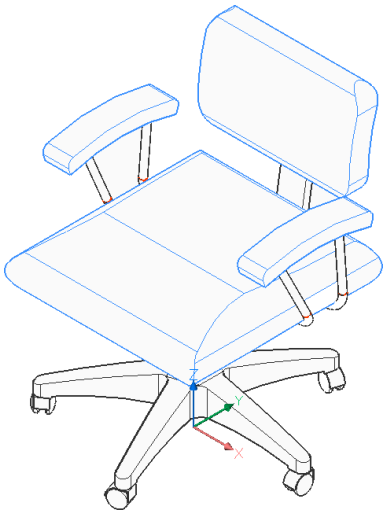
2dwireframe
Default display mode.



Wireframe
Wireframe, which shows all edges.



Hidden
Wireframe with hidden lines removed.



Realistic
Rendered with materials, if available.



Conceptual

Displays 3D entities with smooth shading and the Gooch shading. Gooch shading is a non-photorealistic rendering technique for shading objects, also known as "cool-to-warm" shading.



Shaded

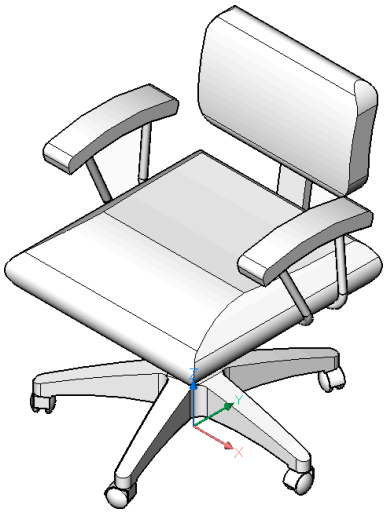
Rendered without materials.



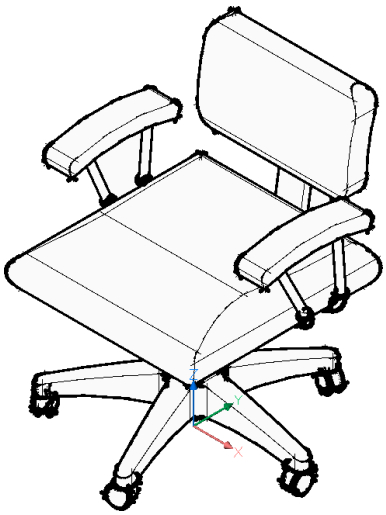
shaded with Edges
Rendered with contrasting edges, without materials.



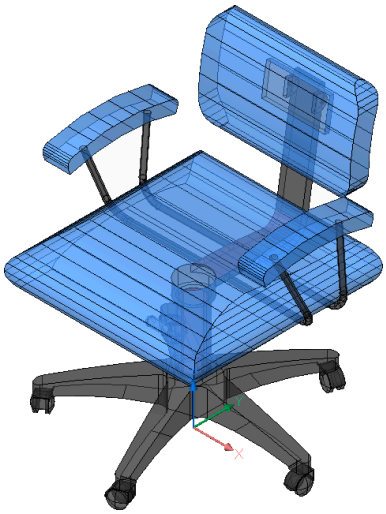
shades of Grey
Rendered in shades of gray, without colors.



Sketchy
Rendered with rough edge lines.



X-ray
Rendered with semi-transparent faces.



Other
Creates a new visual style.

cUrrent
Keeps the current visual style.

24.39 -SHADEMODE command

Specifies the shading style for the SHADE command.

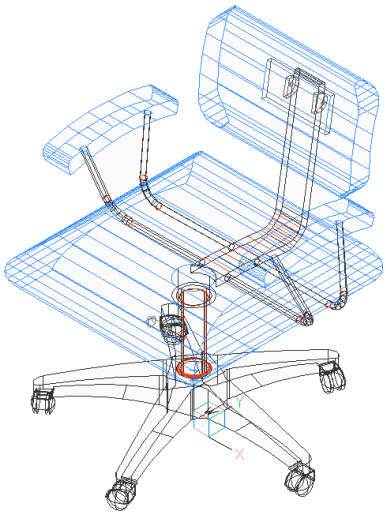
Shape Lite Pro Mechanical BIM

24.39.1 Description

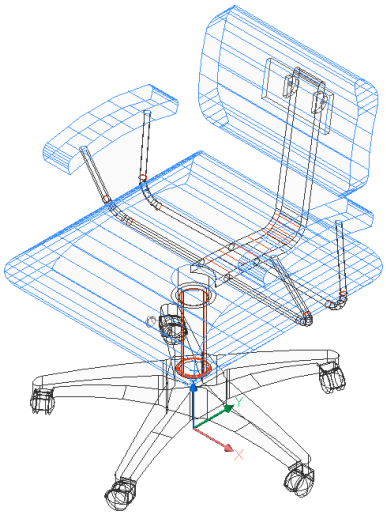
Specifies the shading style for the current drawing to be used by the SHADE command.

24.39.2 Options within the command

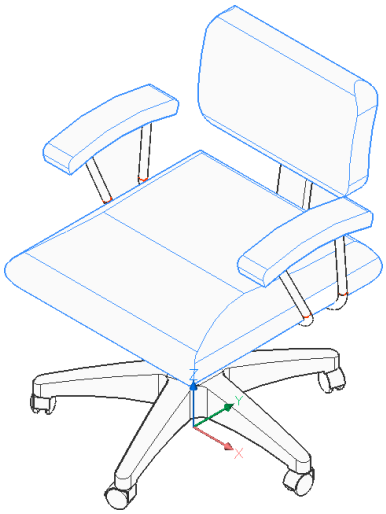
2dwireframe
Wireframe display with no shading.



3d wireframe
Wireframe display with no shading.



Hidden
Wireframe with hidden lines removed.



Flat
Flat shading.



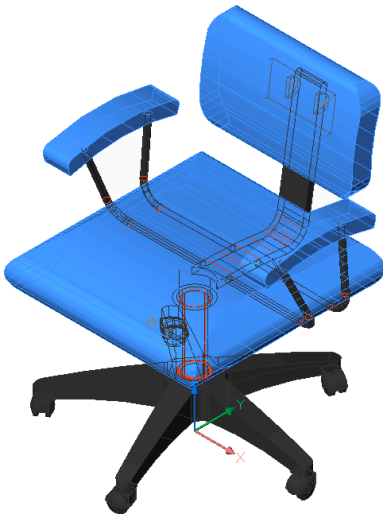
Gouraud
Smooth shading.



fLat+edges
Flat shading with edges highlighted.



gOuraud+edges
Smooth shading with edges highlighted.



24.40 SHAPE command

Places shapes in drawings.

Shape Lite Pro Mechanical BIM

Icon: ✳

Note: Before you can place shapes in drawings, you must first load a SHX shape file with the Load command.

24.40.1 Method

Enter the name of the shape and specifies the insertion point, the scale and rotation angle to place it in the drawing.

Shapes are an early version of blocks that were highly efficient but difficult to code; shapes are rarely used anymore. The SHP format is also used by certain fonts; it contains a flag to distinguish between shapes and fonts.

24.40.2 Options within the command

? to list
Lists the names of shapes loaded in the current drawing. Reports the file name and names of shapes:
Shape file: C:\temp\611.shx
HAIE BOIS
HOTALU BATALU
CLOTUR FROST
RAILS PGA
PGAE PGBR
If no shapes are loaded, reports: “No shapes loaded.”

Note: Shapes can be edited through grips.

24.41 SHEETSET command

Opens the **Sheet Sets** panel.

Shape Lite Pro Mechanical BIM

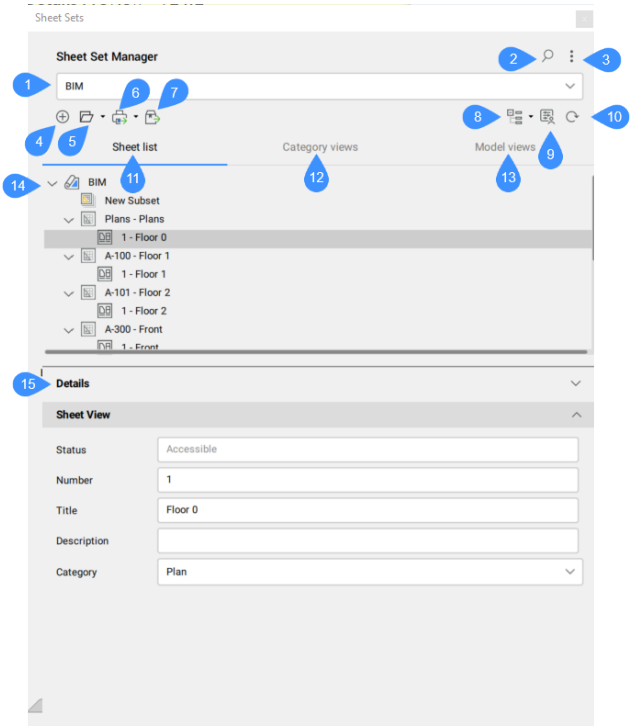
Icon:

Alias: SSM

24.41.1 Description

Opens the **Sheet Sets** panel to display it in the current workspace. The **Sheet Sets** panel appears at the same size and location as it did before it was closed or collapsed. Like any other dockable panel, the **Sheet Sets** panel can be either floating, docked or stacked.

The **Sheet Sets** panel offers a central location to create and to manage sheet sets.



1 Current sheet set name

2 Search

3 Menu

4 Create sheet set

5 Open sheet set

6 Publish

7 eTransmit

8 Sheet selection sets

9 Custom properties

10 Refresh

11 Sheet list

12 Category views

13 Model views

14 Sheet Sets list

15 Details

24.41.2 Current sheet set name

Displays the name of the current sheet set. From the drop-down list you can change the current sheet set.

24.41.3 Search

Opens a search field that allows you to search for specific sheet set, subset or sheet names.

24.41.4 Menu

Opens a menu with a number of options depending on the sheet set panel state:

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.

Export to XML
Exports a sheet set in XML format. Opens the **Export Sheet Set As** dialog box.

Sheet Set Options
Opens the **Sheet Controls** options in the **Settings** dialog box.

Custom Properties
Allows you to add custom properties for sheet sets and sheets. Opens the **Custom Properties** dialog box.

24.41.5 Create sheet set

Opens the **Create New Sheet Set** dialog box.

24.41.6 Open sheet set

Opens the **Select a Sheet Set file** dialog box to select the DST file to be loaded. Click the right-side arrow to open a drop-down menu which contains a list with the recently opened DST files. Click one of these files to load the sheet set into the panel.

Open
Opens the selected DST file from the **Select a Sheet Set file** dialog box.

24.41.7 Publish

Publishes the currently selected sheet set(s), subset(s) or sheet(s) using the **Publish** dialog box. Click the right-side arrow to open a drop-down menu:

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

Publish to PDF
Publishes the currently selected sheet set(s), subset(s) or sheet(s) to PDF using the last used presets.

Publish to Plotter
Publishes the currently selected sheet set(s), subset(s) or sheet(s) using a plotter based on the last used presets.

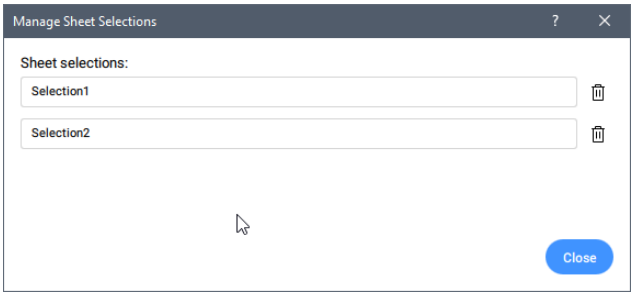
24.41.8 eTransmit

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.

24.41.9 Sheet selection sets

Opens the **Manage Sheet Selections** dialog box. Click the right-side arrow to open a drop-down menu:

Manage...
Opens the **Manage Sheet Selections** dialog box where you can rename or delete Sheet selections.



Create
Saves the Sheet selection with a specified name.

List of existing sheet selection sets
Lists the existing sheet selection sets of the active (selected) sheet set in the panel. Click one of these items to activate the selection set.

24.41.10 Custom properties

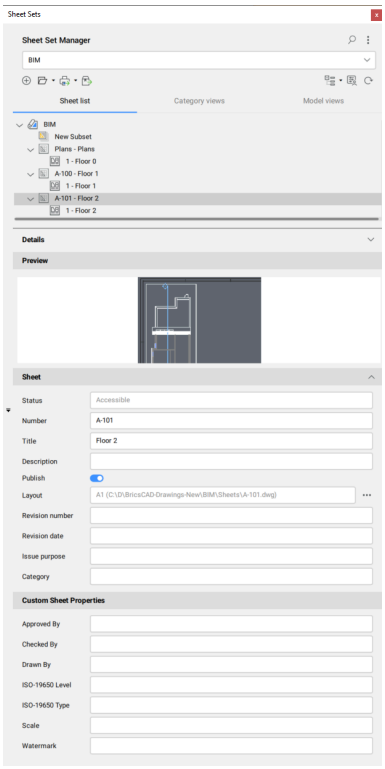
Allows you to add custom properties for sheet sets and sheets. Opens the **Custom Properties** dialog box.

24.41.11 Refresh

Updates all information stored in the sheet set data (DST) file, checks all folders in every resource location for new or removed drawing files, and all expanded drawing files for new or removed model space views.

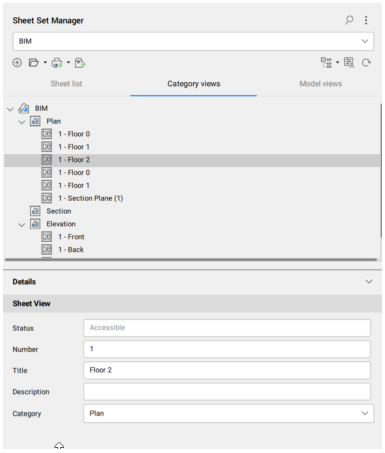
24.41.12 Sheet list

Displays the Sheet Sets in the **Sheet list** mode.

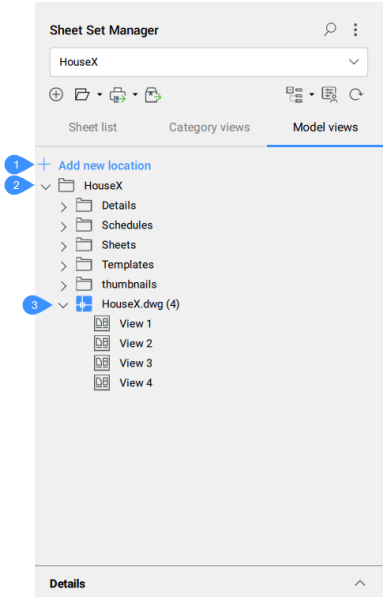


24.41.13 Category view

Displays the Sheet Sets grouped by category.



24.41.14 Model views



1 Add new location

2 Folders

3 Model views

Add new location

Opens the **Browse for Folder** dialog box to choose a folder to be added. Only unique folders can be added.

Folders

Lists all added folders that contain the drawings and their model views. The **Model Views** functionality allows you to browse and select model views (created with the VIEW command) from external drawings and insert them into the Paper Space layout of the current drawing, without opening the original file.

Note: The context menu opened by right-clicking a folder offers the option to **Remove** it from the **Model views** tab.

Model views

Model views (created with the VIEW command) from external drawings can be dragged and dropped into a Paper Space layout of the current drawing.

Note: Double-click a drawing/drawing Model view in the **Model views** tab to open it in the Model Space. Access the **Open**/**Open Read-Only** options from the right-click context menus.

24.41.15 Sheet Sets list

Displays the current sheet set.

When you change the project name in the **BIM Project Browser** panel, the Sheet Set name is updated correspondingly.

A Sheet Set can contain Sheets, Subsets, and Sheet Views.

Right-clickmenusoffer additional tools:

Sheet Set

Close Sheet Set
Removes the Sheet set from the panel.

Export as XML
Opens the **Export Sheet Set As** dialog box.

Resave All Sheets
Saves all sheets.

New Sheet(s)
Displays the **New Sheet(s)** dialog box, which allows you to add new sheets to the current sheet set.

New Subset
Adds a new subset to the current sheet set.

Selection Sets
Opens the **Manage Sheet Selections** dialog box.

Custom Properties
Opens the **Custom Properties** dialog box.

Renumber
Opens the **Renumber Sheet Set** dialog box, which allows you to renumber all sheets in the sheet set.

Plot
Plots the currently selected sheet set(s), subset(s) or sheet(s).

Publish
Opens the **Publish** dialog box.

Export Package (eTransmit)
Opens the **eTransmit** dialog box.

Insert Sheet List Table
Inserts the sheet list table into the current drawing.

Sheet

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.

Remove Sheet
Removes the selected sheet.

Insert Sheet List Table
Inserts a Sheet List Table in the drawing area.

Subset

Rename Subset
Allows you to rename the selected subset.

Remove Subset
Removes the selected subset.

Sheet View

Display View
Opens the drawing file related to the selected Sheet View.

Place Callout Block > Number Bubble
Places a callout block of the corresponding Sheet View.

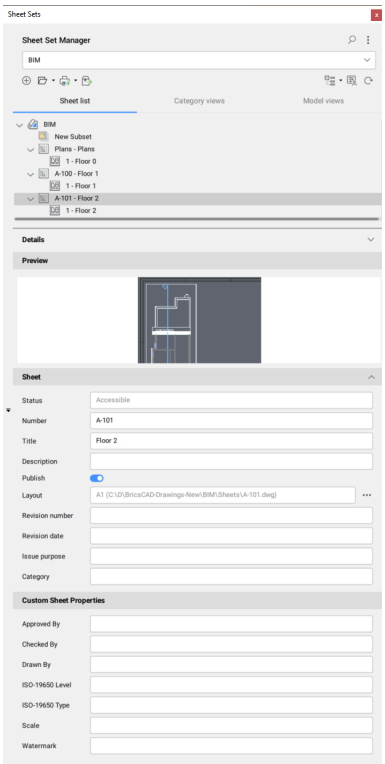
Place View Label Block
Places a view label block of the corresponding Sheet View.

Note:

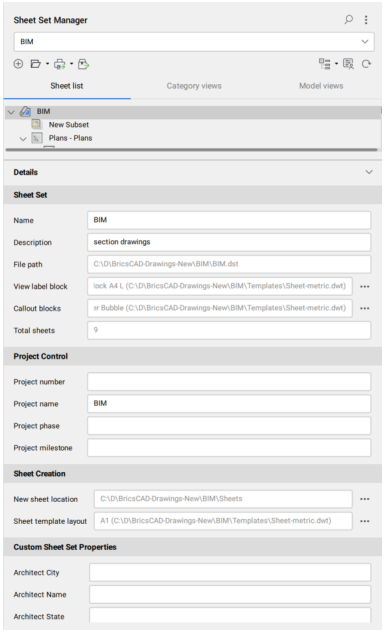
- Double-click a sheet set or a subset to expand/collapse. Double-click a Sheet or a Sheet View to open it.
- Sheet set tree nodes can be reordered using drag and drop.
- The properties and variables of a selected Sheet Set, Subset, Sheet or Sheet View are displayed in the **Details** section.

24.41.16 Details

Displays the details of the selected Sheet Set, Subset, Sheet, or Sheet View. A preview of the selected sheets or subset can be shown or hidden.



In the **Details** > **Sheet Set**, the **File path**, **View label block**, **Callout blocks** and **Total sheets** options are read only, but selectable.



Note:

- Click the **Browse** button (…) next to **View label block** option to open the **Select Block** dialog box.
- Click the **Browse** button (…) next to **Callout blocks** option to open the **Blocks List** dialog box.

24.42 SHEETSETHIDE command

Closes the **Sheet Sets** panel.

Shape Lite Pro Mechanical BIM

24.42.1 Description

Closes the **Sheet Sets** panel to hide it from the current workspace. If the **Sheet Sets** panel is stacked when you close it, the Sheet sets tab or icon is removed from the stack.

24.43 SHELL command

Opens the command prompt window.

Shape Lite Pro Mechanical BIM

24.43.1 Description

Opens the command prompt window and runs other applications.

Note: This is a Windows only command.

24.43.2 Method

There are two methods to use the SHELL command:

- Specify the name of a program to run.
- Press Enter to open the command prompt window.

24.44 SHOWURLS command (Express Tools)

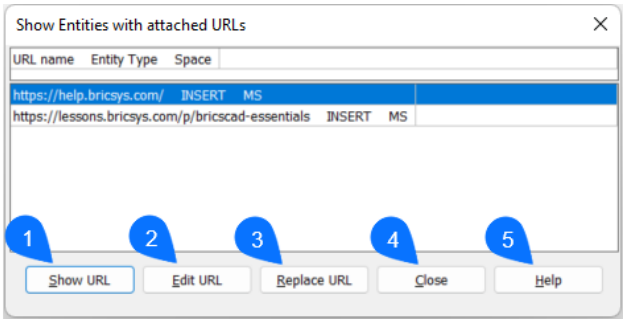
Displays all embedded URL addresses in the drawing and allows you to edit them.

Shape Lite Pro Mechanical BIM

Icon:

24.44.1 Method

Opens the **Show Entities with attached URLs** dialog box that allows you to show, edit and replace URL addresses.



1 Show URL

2 Edit URL

3 Replace URL

4 Close

5 Help

24.44.2 Show URL

Hides the dialog box and highlights the objects linked with the selected URL.

24.44.3 Edit

Allows you to edit the selected URL.

24.44.4 Replace

Allows you to replace the a URL by prompting the old one that has to be replaced and asking for a new one.

24.44.5 Close

Closes the dialog box.

24.44.6 Help

Redirects you to the BricsCAD Help Center.

24.45 SHP2BLK command (Express Tools)

Converts all instances of a selected shape entity with an equivalent block reference.

Shape Lite Pro Mechanical BIM

Icon:

24.45.1 Method

Select a shape entity and enter the name of the replacement block.

Note: It replaces all instances of the shape with instances of the defined block.

24.46 SIGVALIDATE command

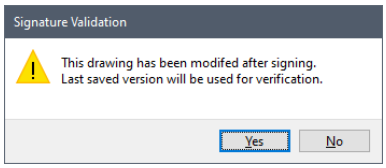
Displays the digital signature for the drawing and its Xrefs.

Shape Lite Pro Mechanical BIM

24.46.1 Method

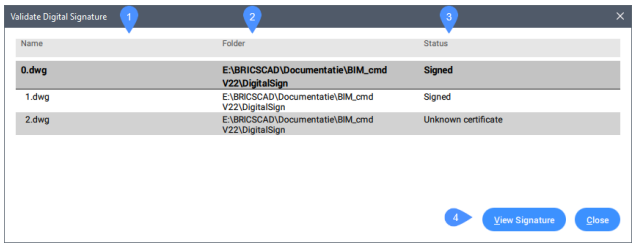
Opens the **Validate Digital Signature** dialog box.

Note: If the current drawing was modified after the last signing, the warning message below will be displayed:



Note: The SIGWARN system variable controls the display of a dialog box with the signature content when a drawing with a digital signature is opened.

Displays the status of all digital signatures for a drawing and its Xrefs.



Note: The first drawing in the list, which is written in a bold large font, is the host drawing.

Name
Displays the drawing file name.

Folder
Displays the drawing folder.

Status
Displays if the drawing is signed with a valid digital signature or not.

Signed
The drawing is signed with a valid digital signature.

Not signed
The drawing is not signed with any digital signature or the modification of the drawing was saved without applying the digital signature again.

Unknown certificate
The digital certificate is not located in the proper location or the digital certificate was revoked by the certificate authority.

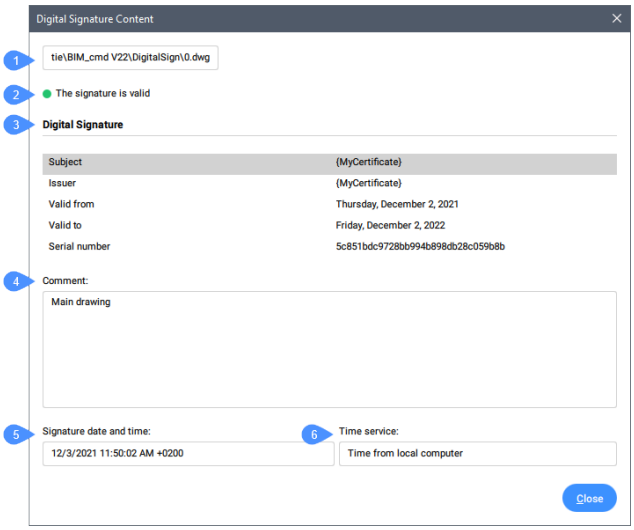
Not found
The drawing lost its initial location.

Invalid signature
The signed drawing was modified in an old BricsCAD version that doesn't support digital signature (V21).

Unloaded
The referenced file is not loaded.

Error
An error occurred during signature validation.

View Signature
Opens the **Digital Signature Content** dialog box:



Drawing identifying
Displays the path of the drawing.

Status of the signature
- Marked in green in case of a valid signature.
- Marked in red in case of an invalid signature.

Digital Signature
Subject: displays the certificate name.
Issuer: displays the author name.

Valid from:displays the date from which the certificate is valid.
Valid to: displays the date until which certificate is valid.
Serial number: displays the certificate serial number.

Comment
Displays the signature comment.

Signature date and time
Displays the date and time when the drawing was signed.

Time service
Displays the chosen time server.

24.47 -SIGVALIDATE command

Displays the digital signature for the drawing via Command line.

Shape Lite Pro Mechanical BIM

24.47.1 Description

Displays in the Command line and Prompt History window the digital signature for the drawing. It lists the **Subject**, **Issuer**, **Expiration date**, and **Serial number**.

24.48 SIMPLIFY command

Simplifies entities.

Shape Lite Pro Mechanical BIM

Icon: 

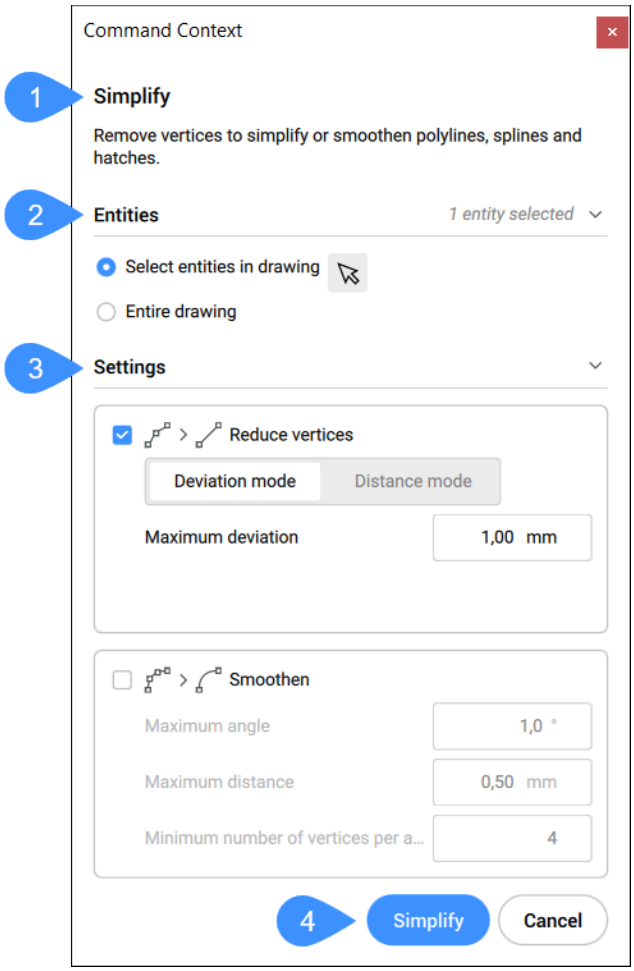
24.48.1 Description

Reduces the number of vertices of polylines, splines and non-associative hatches with geometry boundaries represented by polylines, without changing their general shape.

Note: Simplified entities are easier to manipulate and can significantly reduce file size.

24.48.2 Method

Running the command opens the **Command Context** panel for choosing the command settings.



1 Description

2 Entities

3 Settings

4 Simplify

24.48.3 Entities

Here you can toggle between the options to select the entities that will be simplified.

Select entities in the drawing
This option allows you to select entities in the drawing by clicking on them.

Entire drawing
All entities in the drawing are selected to be simplified.

24.48.4 Settings

Specifies the settings and tolerances on how the entities should be simplified. The two different options, **Reduce vertices** and **Smoothen**, can be toggled on or off.

Note:

- Polylines that contain arcs are never simplified.

- The redundant vertices will be removed even when a polyline has bulges, but the segment with bulges will not be affected.

Reduce vertices
Reduces the number of vertices by combining segments. There are two modes to reduce vertices:

1 **Deviation mode**: removes vertices that fall inside the Maximum deviation to convert segments into a single straight segment. This method is available for lw/2d polylines and splines.

   Maximum deviation
   Specifies the maximum arc's length between each two adjacent vertices and maximum arc's angle between each two adjacent vertices.

2 **Distance mode**: removes vertices, if the variation in angle between the segments is smaller than Maximum angle, to convert segments into a single straight segment, shorter than Maximum distance. This method is available for lw/2d polylines, hatches and 3d polylines.

   Maximum angle
   Specifies the maximum change (angle) in a direction between each two consecutive segments to straighten.

   Maximum distance
   Specifies the maximum distance between non-collinear vertices to straighten.

Smoothen
Controls whether consecutive straight segments (at least three), of which the endpoints lie on an arc, are replaced by a bulged polyline segment.

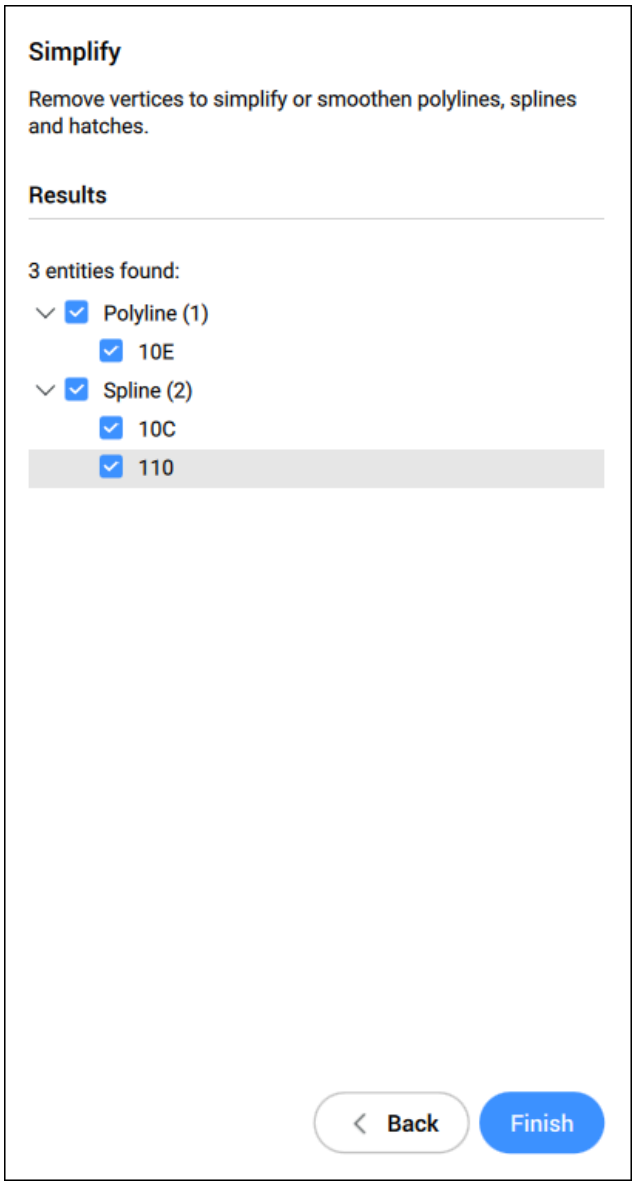
Maximum angle
Specifies the maximum arc's variation in angle between each two adjacent segments.

Maximum distance
Specifies the maximum arc's length between each two adjacent vertices.

Minimum number of vertices per arc
Specifies the minimum number of vertices per arc.

24.48.5 Simplify

After clicking **Simplify**, the panel shows a new screen:



The entities that can be simplified according to the settings are listed here. You can uncheck the entities that should not be simplified. A preview of the simplified entities is shown in the model space when the entities are checked on.

Click **Back** to edit the settings or click **Finish** to simplify the selected entities and close the command.

24.48.6 Options within the Command line

Apply
Applies the settings on the selected entities.

Finish
Closes the command.

Back
Goes back to the options overview.

change Input selection
Changes the selected entities.

Note: The options here are similar to those found in the command context panel's **Entities** section.

change Settings
Allows you to change the settings of the **Reduce vertices** and **Smoothen** modes through the Command line.

Note: The options here are similar to those found in the command context panel's **Settings** section.

24.49 SINGLETON command

Determines whether (or not) multiple copies of BricsCAD can run at the same time.

Shape Lite Pro Mechanical BIM

Note: This command can be entered transparently during commands ('singleton).

24.49.1 Options within the command

Yes
Allows only one copy of BricsCAD to run.

No
Allows two or more copies of BricsCAD to run.

Note: Some third-party applications run only in a single instance of BricsCAD.

24.50 SKETCH command

Creates lines or polylines from a freehand sketch.

Shape Lite Pro Mechanical BIM

Icon:

Alias: FREEHAND

24.50.1 Description

Creates a series of individual lines or a polyline by sketching freehand.



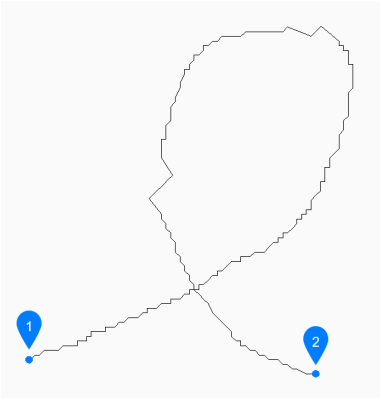
24.50.2 Options within the command

Length of sketch line segments
Specifies the length of the sketch line segments. A small segment length creates a smoother sketch but increases the size of the drawing. The length is saved in the SKETCHINC variable.

Press ENTER or click to toggle
Lowers or raises the pen.

- If the pen is lowered, it automatically sketches as you move it.

- If the pen is raised, you can move the cursor without drawing.



1 Pen up

2 Pen down

Note: You can continue raising and lowering the pen to create unlimited sketches until you press X or Q to end the command.

Erase
Erases sketch lines sequentially by dragging the cursor along the sketch from either endpoint. Click the left mouse button to stop erasing and start sketching.

Connect
Continues to the last sketch point, after raising the pen, by passing the cursor over the end of the previous segment.

Save and exit
Saves the sketch and ends the command. Each continuous sketch is saved as a series of individual lines or a polyline based on the SKPOLY variable.

Quit no save
Exits the command without saving the sketch.

Record (save)
Saves the current sketch without ending the command, allowing you to continue sketching.

24.51 SLICE command

Splits 3D solids and surfaces into two or more entities along a plane.

Shape Lite Pro Mechanical BIM

Icon:

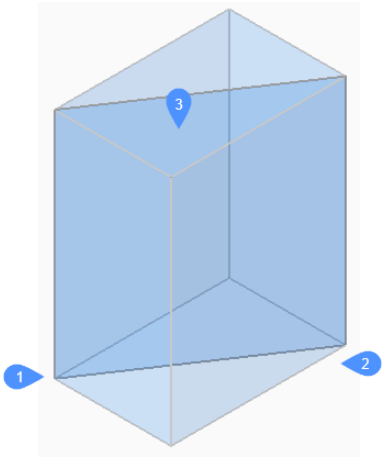
Alias: SL

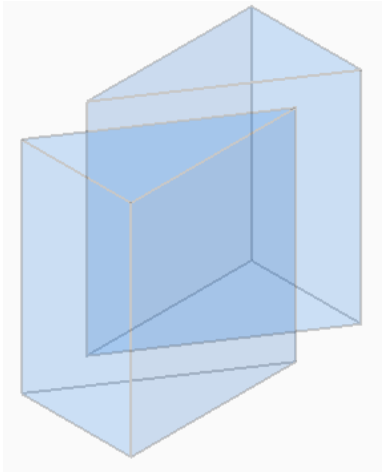
24.51.1 Method

Select one or more 3D solids or surfaces to slice.

Specify the first (1) and the second (2) point on slicing plane. Then, specify a point (3) on the side of the entity you want to keep or choose to keep both. A dotted line from the first point is displayed to easily

visualize the cutting plane. The section plane is perpendicular to the XY-plane of the World coordinate system (WCS).





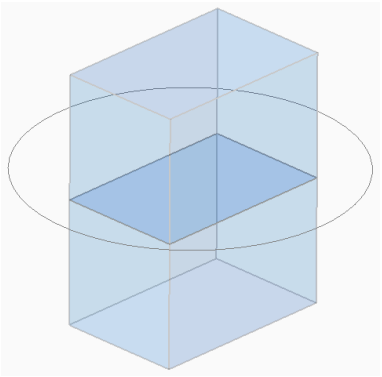
Note:

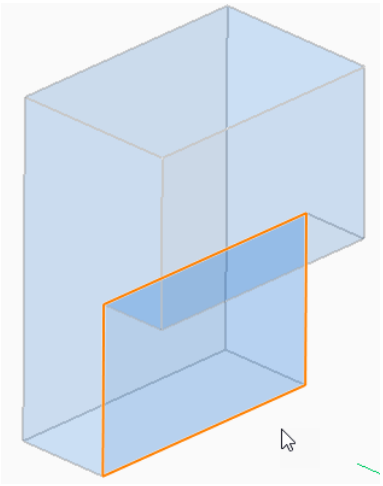
- Ensure that the **Select faces** option of the SELECTIONMODES system variable is enabled to be able to use a planar 3D solid as the slicing plane.
- When the ORTHOMODE system variable is On, you can create straight cutting planes.
- The sliced entities can be manipulated as independent entities.
- This command does not work with 2D regions.

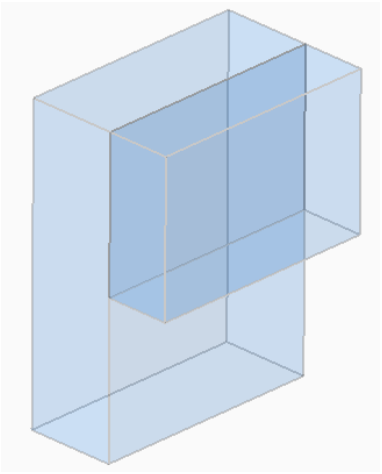
24.51.2 Options within the command

Object

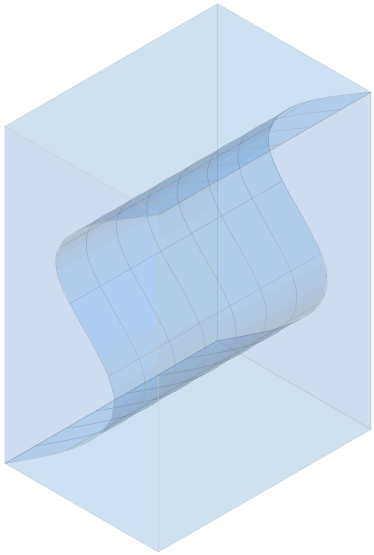
Slices solids and surfaces with an intersecting entity (circle, ellipse, arc, 2D-spline, 2D-polyline, planar 3d Solid face, planar Surface face, or planar region). The entity defines the plane that slices the 3D selected entities.





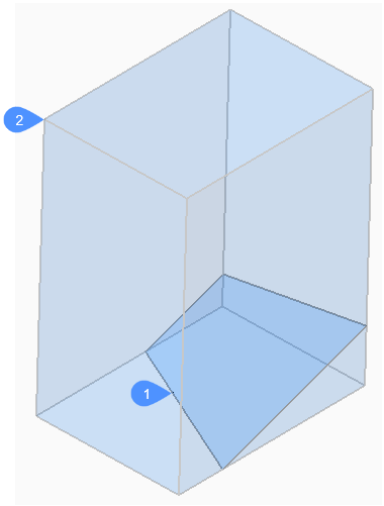


Surface
Slices solids and surfaces with a surface.



Zaxis

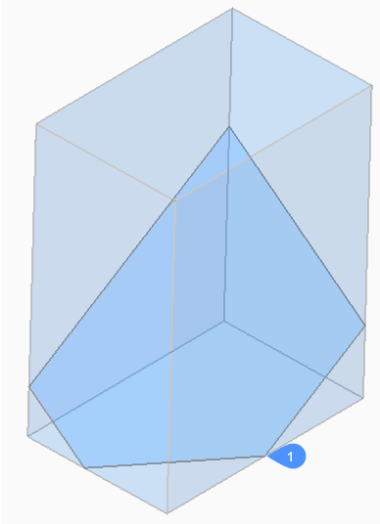
Slices solids and surfaces with a plane perpendicular with a line defined by one point on the plane (1), and a second point on the plane's normal axis (z axis) (2).



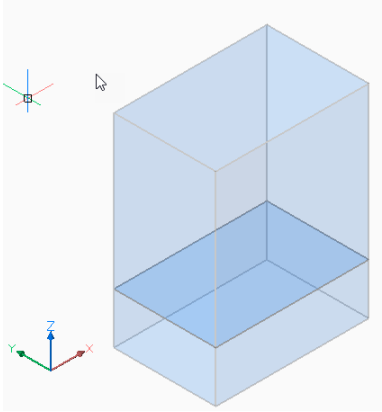
View

Slices solids and surfaces with a plane defined by the current 3D viewpoint.

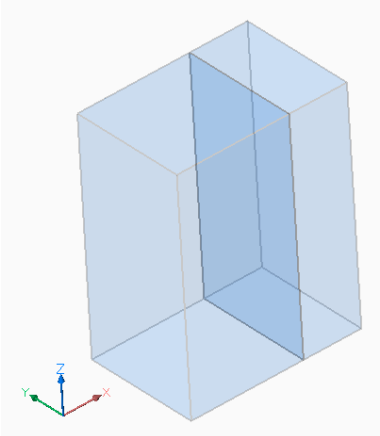
The section plane is perpendicular to the view direction and passes through the picked point.



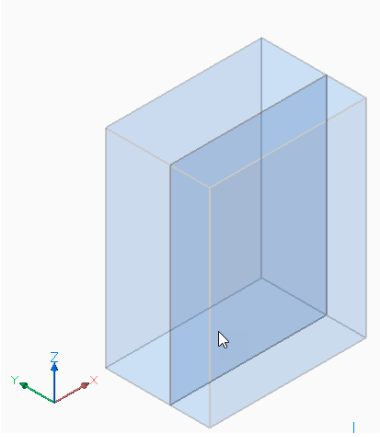
XY
Slices solids and surfaces with a plane parallel to the XY-plane of the current coordinate system.



YZ
Slices solids and surfaces with a plane parallel to the YZ-plane of the current coordinate system.

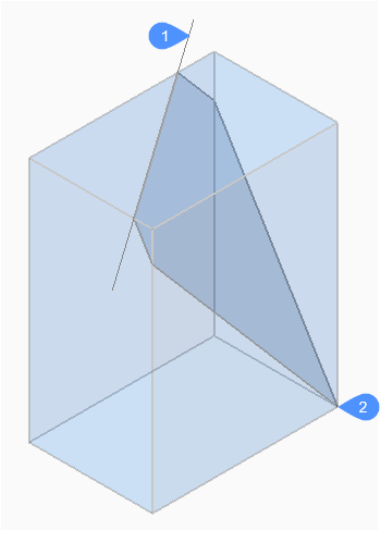


ZX
Slices solids and surfaces with a plane parallel to the ZX-plane of the current coordinate system.



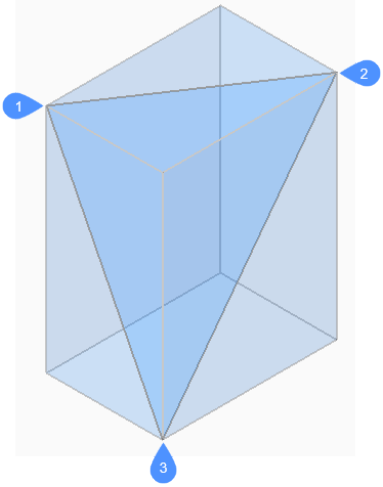
Line-point

Slices solids and surfaces with a plane defined by a linear entity (1) and a point (2).

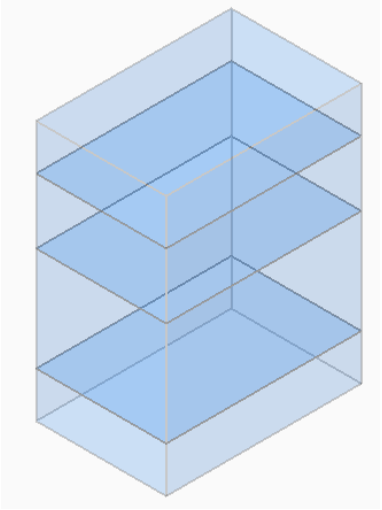


3points

Slices solids and surfaces with a plane defined by three points.



Multislice
Slices solids and surfaces in more than one piece using a face and specifying the distance between slices.



Both
Retains both parts.

24.52 SMASSEMBLYEXPORT command

Searches the assembly structure of the current drawing for 3D solids, mechanical blocks and external references within a component and applies the SMCONVERT and SMEXPORT2D commands.

Shape Lite Pro Mechanical BIM

Icon:

24.52.1 Method

The result of this command is a set of DXF files with unfold information saved in a user-defined output folder and sorted by thickness. Parts processed with command warning or errors are placed in a special folder. An HTML report listing all the solids in the assembly is generated. This indicates their statuses and links to the corresponding DWG and DXF files. The HTML report uses names of mechanical solids instead of their handles when possible.

Note: SMASSEMBLYEXPORT can be applied to mixed assemblies containing both sheet metal and non-sheet metal parts. It quickly sorts out non-sheet metal parts from sheet metal parts.

The solid classification is as follows:

- Sheet Metal – solid is a sheet metal part.
- Poor Sheet Metal – solid looks like a sheet metal design, user assistance and rework are needed.
- Not Sheet Metal – solid is not recognized as a sheet metal design.

Note: SMASSEMBLYEXPORT can be applied to simple drawings with solids, to assemblies created in BricsCAD and to drawings imported with the Communicator for BricsCAD®.

Note: For optimal processing, set the IMPORTPRODUCTSTRUCTURE system variable to 2, otherwise mechanical components will be exploded to solids, increasing processing time.

Note: For optimal performance set the Visual Style to 2dWireframe.

24.52.2 Options within the command

Output folder
Specifies the output folder.

bend Table
Assigns a bend table used during the SMEXPORT2D call in the output process.

Dxf version
Specifies the DXF version.

24.53 SMBEAD command

Creates associative bead features along a curve lying on the flange.

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Icon: 

24.53.1 Description

Creates associative bead features on sheet metal parts from 2D profiles. The feature is updated automatically when its defining profile is changed.

The difference with linear form features inserted from the library by BMINSERT command is that bead features created by the SMBEAD command can have an arbitrary trajectory.

24.53.2 Method

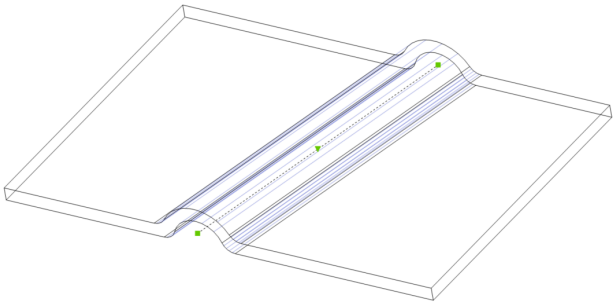
Select the flange face and the control curve to create the bead, using the default profile radius and round radius values.

Note:

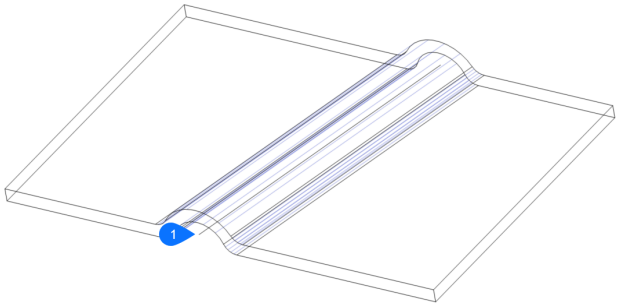
- The default value of the profile radius is defined with the SMDEFAULTRIBPROFILERADIUSVALUE system variable.
- The default value of the round radius is defined with the SMDEFAULTRIBROUNDRADIUSVALUE system variable.

24.53.3 Options within the command

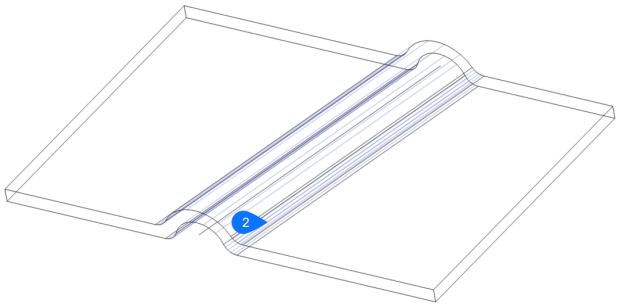
Select control curve
Selects a control curve along which the bead will be created. The selected curve can be a line, polyline, circle, arc, ellipse, elliptical arc or a spline. The 2D curve can be open or closed but it cannot intersect itself or other form features.



Profile radius
Sets the radius of the bead profile (1).



Round radius
Sets the radius of the smoothing fillets (2).



24.54 SMBEND command

Converts hard edges (sharp edges between flange faces) or junctions into bends.

Shape | Lite | Pro | Mechanical | BIM

Icon:

24.54.1 Method

Select hard edges, junctions, flanges or 3D solids.





24.54.2 Options within the command

Entire Model
Detects and converts hard edges or junctions in the entire model.

24.55 SMBENDCREATE command

Converts hard edges (sharp edges between flange faces) or junctions into bends.

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24.55.1 Description

This command is obsolete. Use the SMBEND command instead.

24.56 SMBENDSWITCH command

Converts bends to lofted bends.

Shape Lite Pro Mechanical BIM

Icon:

24.56.1 Description

Allows you to select bend faces which will be transformed into lofted bends.

24.57 SMCONVERT command

Automatically recognizes sheet metal features.

Shape Lite Pro Mechanical BIM

Icon:

24.57.1 Description

Automatically recognizes flanges, bends, lofted bends, ribs, form features and holes for selected 3D solids.

24.57.2 Options within the command

Entire Model

Applies the command to the entire model.

Note: All the solids in the model are analyzed and if possible, converted to Sheet Metal bodies. The command recognizes sheet metal features (such as flanges and bends) in the input geometry. The body thickness of the sheet metal drawing is the criteria in case multiple solids are selected. Therefore, selected bodies must have the same thickness, otherwise an error message displays in the command window.

24.58 SMDELETE command

Removes features from sheet metal elements.

Shape Lite Pro Mechanical BIM

Icon:

24.58.1 Description

Removes a bend or a junction by restoring a hard edge between two flanges. This command can also remove a flange with all the bends adjacent to it and it can remove miter by restoring geometry being cut by the feature.

The adjacent flanges are extended up to a junction configuration with the flange being deleted.

24.58.2 Method

There are two methods to replace features by hard edges:

- Select feature faces.
- Select thickness faces.

24.59 SMDISSOLVE command

Removes the sheet metal data from the selected features.

Shape Lite Pro Mechanical BIM

Icon:

24.59.1 Description

Dissolves sheet metal feature on 3D solid faces. Dissolved features are no longer listed in the Mechanical Browser. Faces of dissolved features can be moved, rotated or push-pulled freely.

24.59.2 Method

There are two methods to remove sheet metal data from features:

- Select faces on model.
- Remove from the entire drawing.

24.60 SMEXPLODE command

Converts bends, forms, hems, jogs, junctions and tabs into their possibly primitive features and removes the original feature.

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Icon:

24.60.1 Method

The feature is either decomposed into some primitives (i.e. a form feature into bends and flanges), or separated into several ones of the same type (i.e. a bend with hole into two bends):

- Bend containing several connectivity components into a list of bend features;
- Junction with several components into several junction;
- Form into flanges and bends;
- Jog into flanges and bends;
- Hem into flanges and bends;
- Tab into multiple junctions;

24.61 SMEXPORT2D command

Exports the unfolded representation of a sheet metal body.

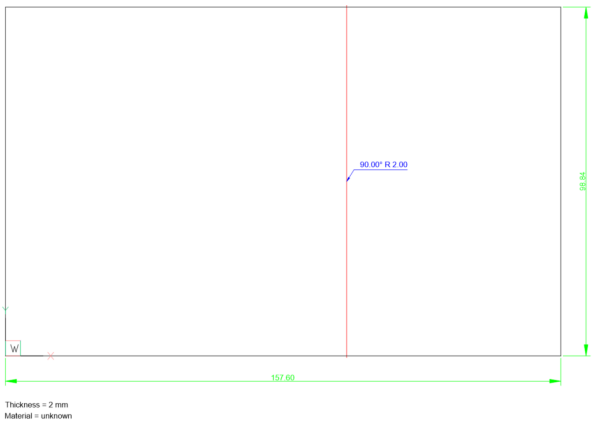
Shape Lite Pro Mechanical BIM

Icon: DXF

24.61.1 Description

Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

Displays the Save unfolded 2D entities dialog box. Here you can save the 2D profile file.



24.62 SMEXPORTOSM command

Exports a sheet metal solid to an Open Sheet Metal (.osm) file format.

Shape Lite Pro Mechanical BIM

Icon: OSM

24.62.1 Description

Reinterprets the supplied sheet metal model as an OSM part to be saved in an *.osm document.

The command is side-sensitive. It unfolds and scales the unfolded planes for each feature and then it connects these plane parts in accordance with the original model in the resulting *.osm document.

The .osm part is saved as mm- or inch-based, depending on the value of the document MEASUREMENT system variable. Document INSUNITS are considered for proper scaling.

The custom properties of all mechanical objects are imported into the OSM document and the OSM tags.

Note: You can set the accuracy of the approximation and the minimal edge length in the units of the document.

24.62.2 Options within the command

add Decorations
Exports non-sheet metal parts in the same .osm part.

add 2D entities
Exports 2D entities in the same .osm part.

set Layer
Specifies the layer where the 2d entities will be stored.

24.63 SMEXTRUDE command

Creates sheet metal part by extruding a planar polyline.

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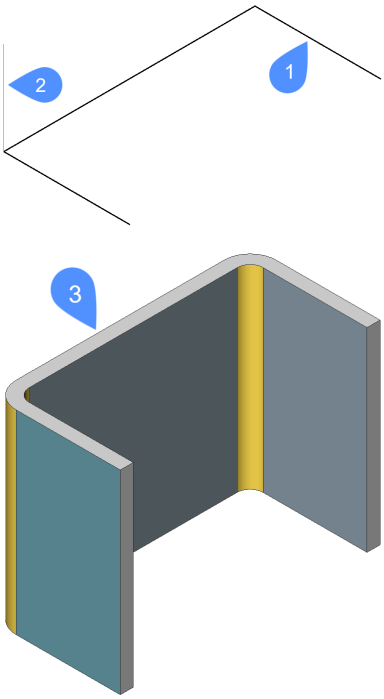
Icon:

24.63.1 Description

Creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.

24.63.2 Method

Select a planar polyline and specify the height of extrusion.



1 Planar polyline

2 Extrusion direction

3 Extruded polyline

The thickening direction of the extruded surface is applied for each flange separately, to preserve the overall dimensions of polyline.

The overall dimensions of the Sheet Metal part match the dimensions of the extruded polyline.

24.64 SMFLANGE command

Creates a base (initial) flange.

Shape Lite Pro Mechanical BIM

Icon:

24.64.1 Description

Creates a base (initial) flange of a sheet metal part from a closed planar profile.

Note: A base flange is created for each selected closed polyline or region. In the **Mechanical Browser** a body and a flange are added for each entity.

24.64.2 Options within the command

Up
The flange is extruded on top of the base entity.

Middle
The flange is extruded on both sides of the base entity.

Down
The flange is extruded below the base entity.

Note: If the **Hot Key Assistant** is toggled ON, you can cycle the various option by pressing the Ctrl key.

Accept model
Accept the current extrusion.

24.65 SMFLANGEBASE command

Creates a base (initial) flange.

Shape Lite Pro Mechanical BIM

24.65.1 Description

This command is obsolete. Use the SMFLANGE command instead.

24.66 SMFLANGEBEND command

Allows to bend an existing flange along a line, obeying the k-factor for given bend radius.

Shape Lite Pro Mechanical BIM

Icon: 

24.66.1 Options within the command

New line
Allows to draw a line to define the bend location.

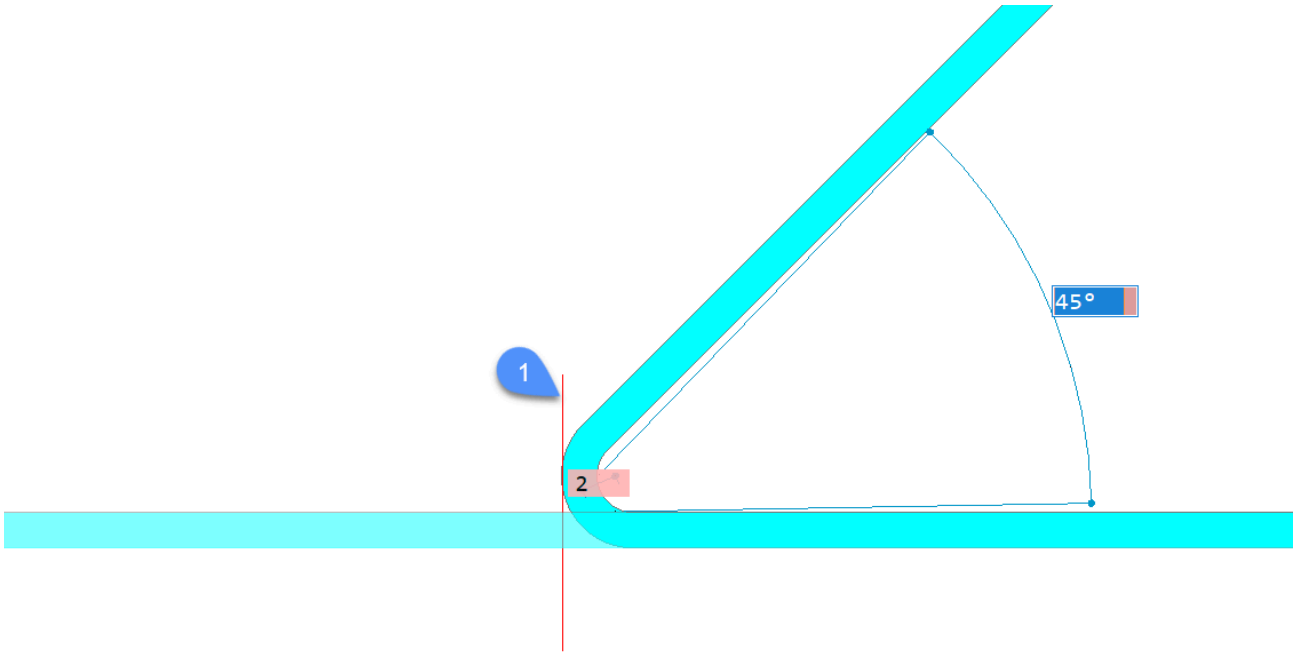
Angle
Specifies the bend angle.

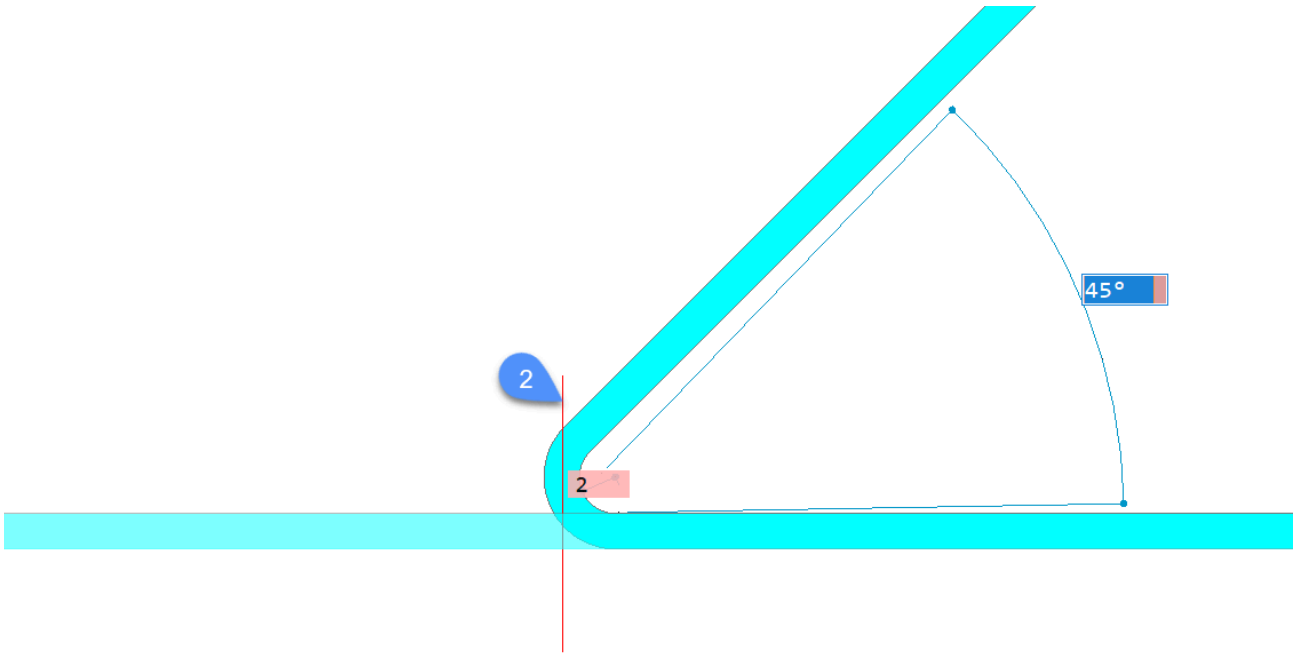
Radius
Specifies the bend radius.

Note: The default bend radius is defined by the Modeling/Bend Radius property in the Mechanical Browser for Sheet Metal.

Switch sides
Allows to move the part of the flange on the other side of the line or edge.

Toggle bend extents
Toggles between keeping the bend extents (1) or not (2). Default is not.


45°
1
2


45°
2
2

ACcept
Creates the bend at the current angle and radius.

24.67 SMFLANGECONNECT command

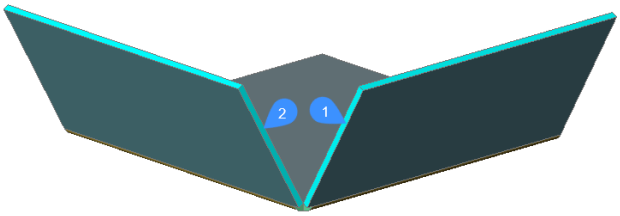
Closes gaps between two arbitrarily oriented non-coplanar flanges.

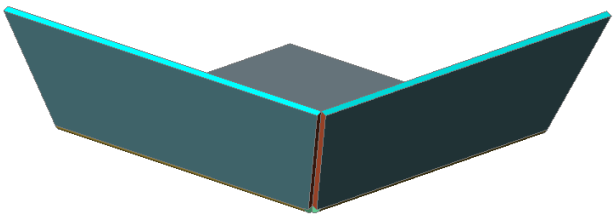
Shape | Lite | Pro | Mechanical | BIM

Icon:

24.67.1 Method

Select planar thickness faces of two flanges to connect them.





Note: If SELECTIONPREVIEW system variable is 2 or 3, thickness faces of flanges are highlighted. Press the TAB key to select obscured faces.

24.68 SMFLANGECONTOUR command

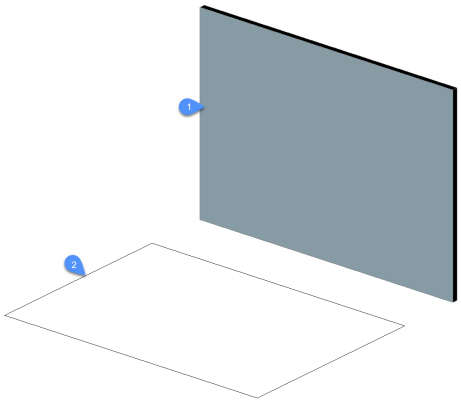
Creates a new flange from a closed contour and attaches it to an existing sheet metal part.

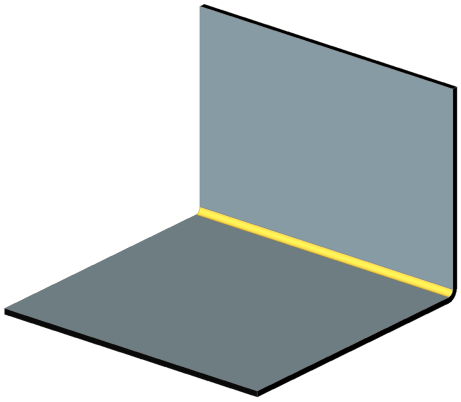
Shape | Lite | Pro | Mechanical | BIM

Icon:

24.68.1 Description

Extrudes a flange from a contour and connects it with selected flanges via bends.





24.69 SMFLANGEEDGE command

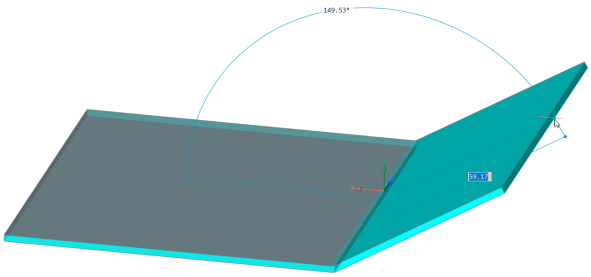
Creates flanges to a sheet metal part.

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Icon:

24.69.1 Description

Creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange or lofted bend.



24.69.2 Options within the command

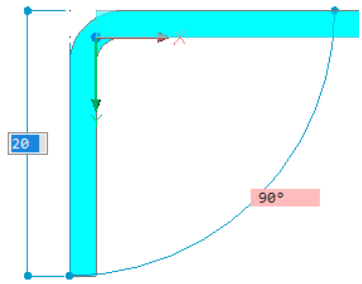
Dynamic dimensions

Creates a new flange using the dynamic dimensions of the height and angle of the new flange or by positioning the location.

Note: Use the Tab key to switch between the height and angle field.

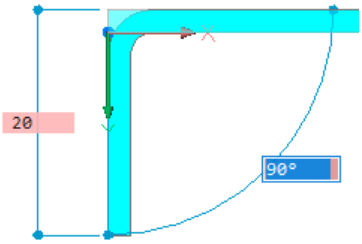
Material Outside

Creates a flange shifted by one material thickness to the outside.



material Inside

Creates a flange preserving the external dimensions of the parent flange.



Angle

Allows you to first define the angle of the flange, then the height.

Length

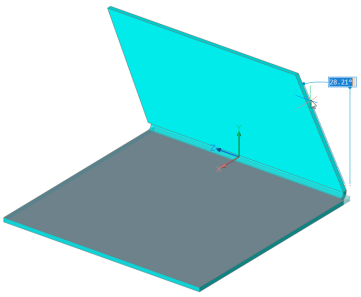
Allows you to first define the length of the flange, then the angle.

Radius

Defines the radius of the bend connected to the new flange.

Taper Angle

Creates a flange with one or two tapered side faces.

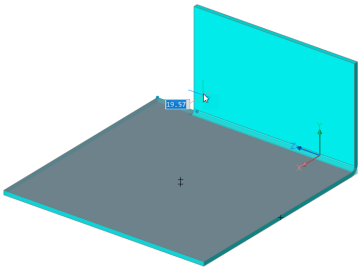


Note: This option is not available if multiple edges are selected.

Back
Goes back to the previous prompt.

Skip
Keeps the flange side orthogonal instead of tapering it.

Width
Allows to define a different width than the length of the selected edge. By default, the flange width equals the length of the selected edge.



Note: This option is not available if multiple edges are selected.

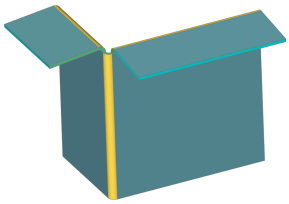
Back
Undoes the current width procedure.

Skip
Keeps the side edge of the flange at the end point of the edge.

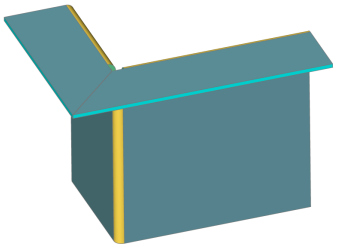
Toggle connection
Toggles between different connection options when creating several flanges simultaneously.

Note: The toggle connection is only available when at least two adjacent edges are selected.

No connection



With connection



Note: Use the Ctrl key to cycle through the options. Make sure that the Hot Key Assistant (HKA field in the Status bar) is ON.

Note: If TIPS = ON, the currently selected option is indicated in the Tips widget at the bottom of the screen.

set Direction
Defines the angle of the flange by specifying a direction to be parallel to.

2Points
Defines the direction by specifying two points.

Object
Defines the direction by selecting an axial entity (for example, linear edges or planar faces).

Last
Uses the last used direction.

View
Uses the Z direction of the view.

Xaxis
Uses the X axis as direction.

Yaxis
Uses the Y axis as direction.

Zaxis
Uses the Z axis direction, if suitable.

24.70 SMFLANGEROTATE command

Rotates a selected flange of a sheet metal part.

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24.70.1 Description

This command is obsolete. Use the SMROTATE command instead.

24.71 SMFLIP command

Swaps sides of a selected flange.

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Icon: 

Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. Optionally shifts the flange over the thickness of the sheet metal part.

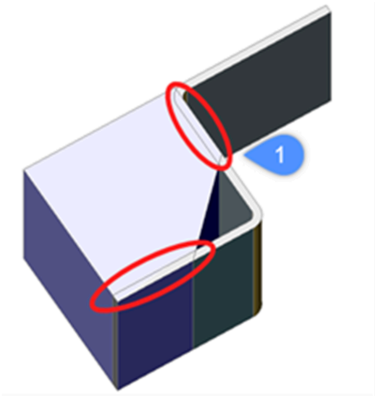
24.71.1 Method

Select a 3D solid or a flange face or press Enter to flip sides for all flanges.

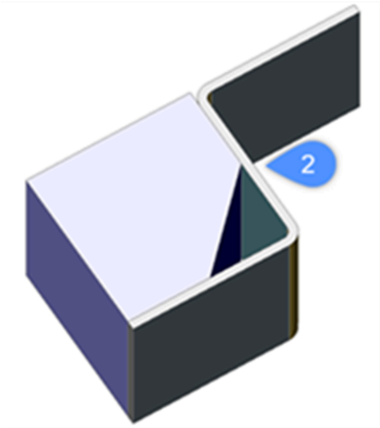
24.71.2 Options within the command

Flip reference side only
Turns OFF the shift of flanges by thickness, in this mode reference sides for selected flanges are just swapped.

Note: Changing reference faces with shift by thickness might be necessary if SMEXTRUDE command creates flanges which collide with another solid.



1. Collisions between a solid and sheet metal flanges.



2. Collisions solved after applying SMFLIP.

Entire Model
Applies flip operation to all flanges in the drawing.

24.72 SMFORM command

Converts a selected set of faces to a form feature.

Shape Lite Pro Mechanical BIM

Icon:

24.72.1 Method

This command converts a selected set of faces to a form feature or explodes form features into flanges and bends:

In Create mode, the command allows you to define a form feature by selecting the faces manually.

Note: This can be necessary if automatic recognition by SMCONVERT does not produce the expected result.

For example: if you insert and dissolve the **Cardguide** form feature from the library, SMCONVERT recognizes two forms features because their geometry is separated. A manual selection of the form feature faces when executing SMFORM solves the problem.

Note: Form features created by SMFORM have the same set of basic operations as if they were created by BMINSERT or recognized by SMCONVERT.

In Explode mode, the command dissolves the form feature and tries to recognize its geometry as flanges and bends. This can be helpful if SMCONVERT incorrectly recognizes the design as a form feature.

24.72.2 Options within the command

Create new form feature
Creates new form features.

Explode form feature
Explodes existing form features into flanges and bends.

Note: At least one face of each form feature to be exploded must be selected.

COnvert to form
Allows converting bead and bevel features in forms.

24.73 SMHEM command

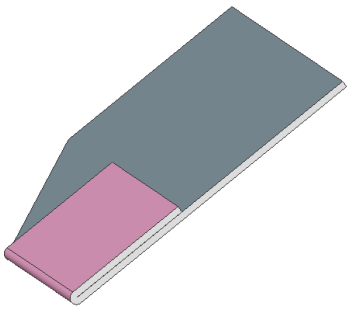
Creates one or more hem features on a sheet metal part by pulling one or more edges of an existing flange.

Shape Lite Pro Mechanical BIM

Icons: 

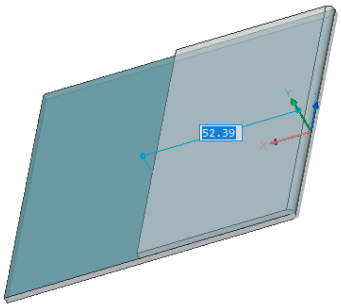
24.73.1 Method

By default, the command creates closed hem.



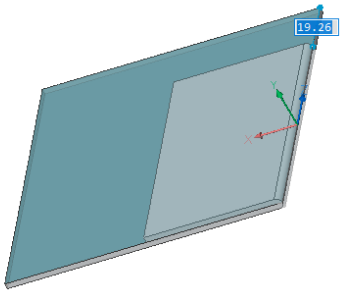
24.73.2 Option within the command

Length
Input the length of the hem.

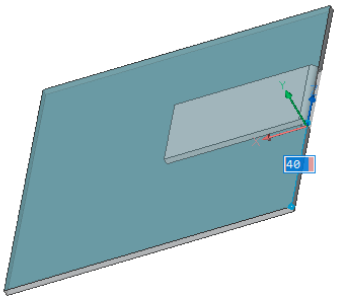


Width
By default, the hem width is equal to the length of the selected edge. The **Width** option defines a different width.

Note: The **Width** option is not available if multiple edges are selected.

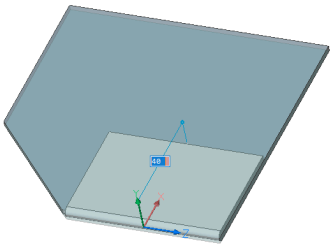


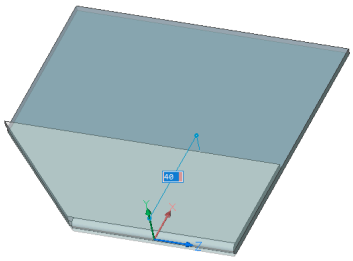
Skip
Keeps the distance value for the first side edge of the flange at the start point of the edge and switch the dynamic field to define the offset from the end point of the edge.



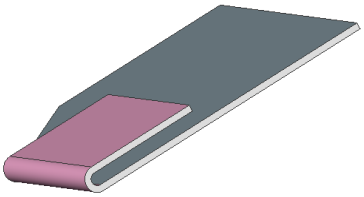
Back
Undoes the current **Width** procedure.

Toggle tapering
Toggle how a new hem fits the border of an existing flange. Two variants are possible:
The option only works if the edges of a flange at the corner are not at 90°.



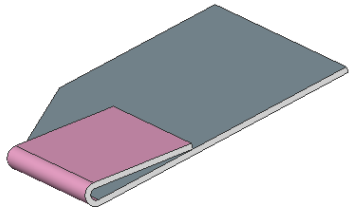


open Hem
Creates an open hem.

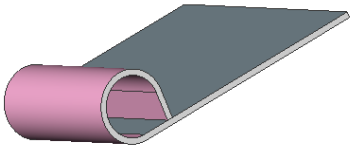


Radius
Defines the radius of the hem.

Teardrop
Creates a teardrop shape hem.



Round
Creates a round shape hem.



24.74 SMHEMCREATE command

Creates one or more hem features on a sheet metal part by pulling one or more edges of an existing flange.

Shape Lite Pro Mechanical BIM

24.74.1 Method

This command is obsolete. Use the SMHEM command instead.

24.75 SMIMPRINT command

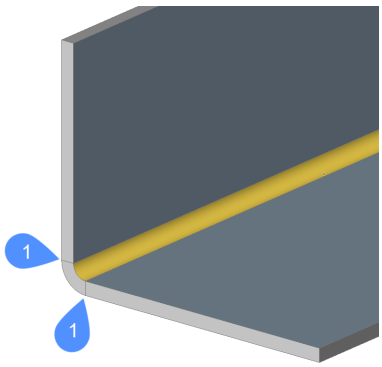
Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.

Shape Lite Pro Mechanical BIM

Icon:

24.75.1 Description

Imprints edges on thickness faces to separate them into areas, which can be reworked automatically.

	1 Imprinted edges

24.76 SMJOG command

Creates a jog on a flange or on cylindrical lofted bends that avoid the collision with the other extent and allowing to do the smallest cuts.

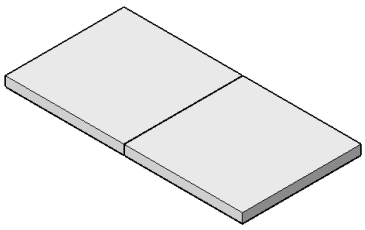
Shape Lite Pro Mechanical BIM

Icon:

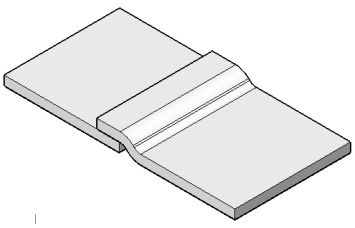
24.76.1 Method

Jog on a flange

Select an edge of a flange:



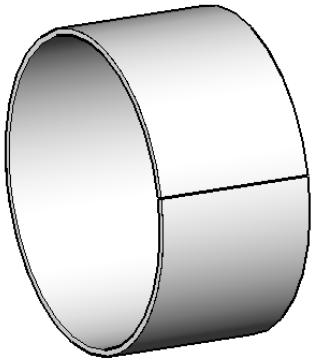
When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.



Note: There is no collision on the other part.

Jog on cylindrical lofted bends

Select an edge of the cylindrical lofted bend.

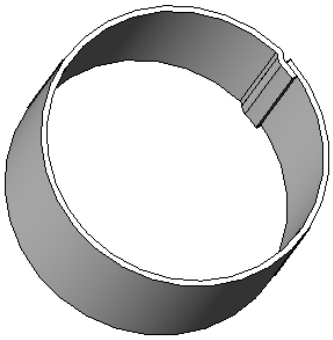


When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.



Note: There is no collision on the other part.

Optionally, jog can be built on internal edges:



Note: If the FEATURECOLORS system variable is ON, the jogs will be displayed in the color set by SMCOLORJOG system variable.

24.76.2 Option within the command

Angle
Sets the jog angle; the default angle is 45 degrees.

Length
Sets the length of the jog. When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.

Height
Sets the height of the jog.

Radius
Sets the radius for the jog.

Note: The starting point to measure the above dimensions is on the selected edge.

24.77 SMJUNCTION command

Creates a junction feature on hard edges (sharp edges between flange faces) and bends or curved hard edges between two bends (or lofts).

Shape Lite Pro Mechanical BIM

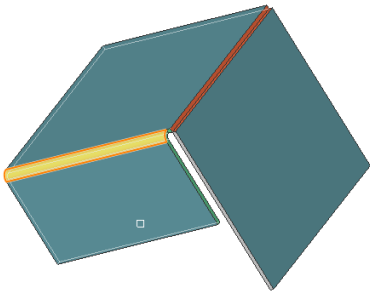
Icon:

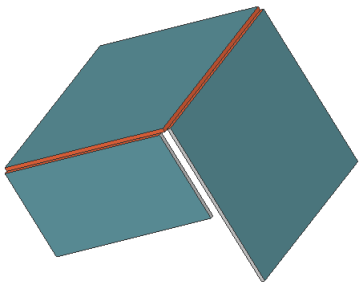
24.77.1 Description

The "Heal coincident junction" setting (SMJUNCTIONCREATEHEALCOINCIDENT system variable) controls how BricsCAD recognizes junction designs with coincident faces and how they are converted to regular junctions.

24.77.2 Method

Select flanges, hard edges, bends or 3D solids. You can also select "curved" edges between flanges and bends, between flanges and lofted bends, between two bends, or between two lofts.





24.77.3 Option within the command

Entire model
Analyzes the entire model to detect hard edges. A curved hard edge into a curved tabbed connection of a flange and a bend (or lofted bend).

24.78 SMJUNCTIONCREATE command

Creates a junction feature on hard edges (sharp edges between flange faces) and bends or curved hard edges between two bends (or lofts).

Shape Lite Pro Mechanical BIM

24.78.1 Description

This command is obsolete. Use the SMJUNCTION command instead.

24.79 SMJUNCTIONSWITCH command

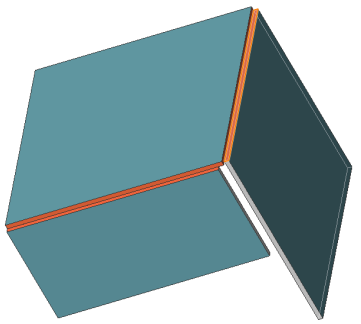
Allows changing a symmetrical junction feature to one with overlapping faces.

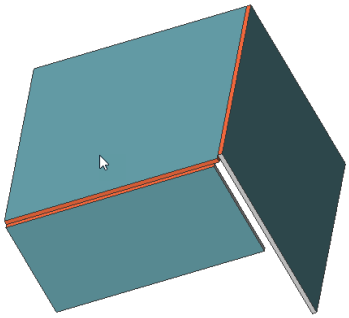
Shape Lite Pro Mechanical BIM

Icon:

24.79.1 Method

Select the junction face (left side picture) and see how the junction is transformed (right side picture).





24.80 SMLOFT command

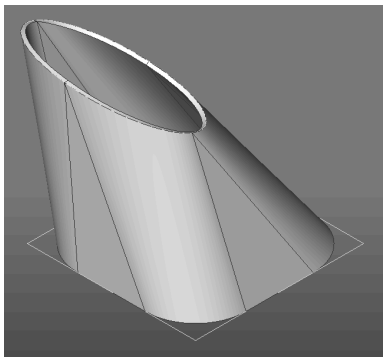
Creates sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.80.1 Description

Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.



24.80.2 Options within the command

fillets Radius

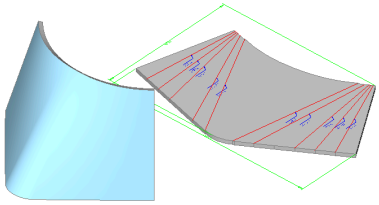
Sets radius of fillet which will be created on straight lines connections. A radius can be specified or the minimum radius can be calculated regarding the thickness of the model by pressing enter.

Thickness

Sets model thickness.

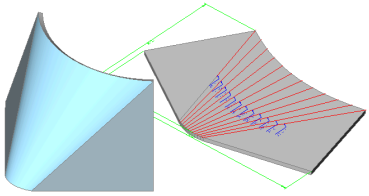
Single bend

Creates only lofted bend feature without any flanges.



Flanges with bends

Creates a flange feature for each straight segment of the selected 2D entities or a lofted bend feature for curved segments.



Alignment
After setting all options, the alignment of the metal sheet loft can be chosen.

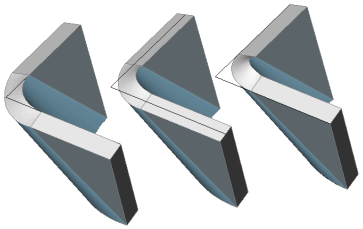
Note: Hit the CTRL key to adjust the alignment: Inside, Both Sides or Outside.



thicken Inside
Thickening direction is selected that way that 3D part is inside the selected 2D entities.

Both sides
Thickens at both sides of the selected 2D entities.

thicken Outside
Thickening direction is selected that way that 3D part is outside the selected 2D entities.



24.81 SMPARAMETRIZE command

Creates a consistent set of 3D constraint for a selected sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.81.1 Description

Creates a consistent set of 3D constraints for a selected sheet metal part and recognizes a rectangular array of holes on flanges. The command takes into account implicit constraints from sheet metal features and ensures that the constraint system is not overdefined.

For the best results, the sheet metal part should have a complete set of features including flanges, bends, junctions and bend & corner reliefs.

Constraints can be edited in the **Mechanical Browser** panel.

24.82 SMQUICK command

Suggests the optimal set of splits/junctions/bends for a shelled solid.

Shape Lite Pro Mechanical BIM

Icon:

24.82.1 Method

This command can be used on 3D Solids and on Sheet Metal solids.

If a 3D Solid is selected, this command can be used to:

- Create shelled solids.
- Create reliefs.
- Convert hard edges into junctions and bends.

If a Sheet Metal Solid is selected, this command recognizes its features and creates reliefs, junctions, and bends.

24.82.2 Options within the command

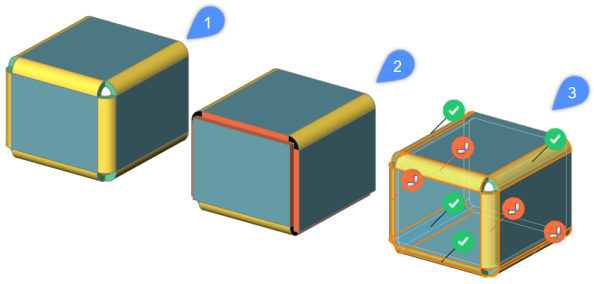
Enter shell offset distance
Specifies a distance.

Note: If a positive distance is entered, the body shells outwards. If a negative distance is entered, the body shells inwards.

Note: If the shell offset distance is larger than 3D solid, the command fails.

Skip
Skips the creation of shelled solid and acts similar to SMCONVERT command.

Select optimization mode
Select a mode to optimize the model.



Off (1)
Creates bends for all hard edges.

Auto (2)
Creates bends, junctions and split.

Interactive (3)
Gives suggestions for bend, junctions and split using a widget interface. Choose to apply the suggested configuration or change the suggestions.

24.83 SMRELIEF command

Creates proper corner and bend reliefs.

Shape Lite Pro Mechanical BIM

Icon: 

24.83.1 Description

Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge. Command supports the creation of reliefs on wrong bends.

The command automatically decides whether a relief must be created for a given bend. The size is either automatically chosen by default from the **Mechanical Browser for Sheet Metal** in the drawing or can be entered in the Command line.

In order to force the creation of bend relief for given bend, select the face of desired bend. In this case a bend will be created, even if the command decides a bend relief is not needed (i.e., it detects there are no clashes or stretches of the material).

24.83.2 Options within the command

Entire model
Creates reliefs on all hard edges, bends and corners in the entire model.

force Bend reliefs
Forces bend relief creation on bends.

Auto
The command automatically detects bend radii and adjusts the relief size accordingly.

24.84 SMRELIEFCREATE command

Creates proper corner and bend reliefs.

Shape Lite Pro Mechanical BIM

24.84.1 Description

This command is obsolete. Use the SMRELIEF command instead.

24.85 SMRELIEFSWITCH command

Converts corner reliefs.

Shape Lite Pro Mechanical BIM

Icons: 

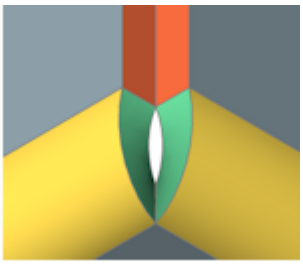
24.85.1 Description

Converts corner reliefs to circular, rectangular or V-shaped reliefs. Converts bend reliefs to smooth, round, rip or rectangular bend reliefs. This command also allows to change the parameters of existing corner reliefs.

24.85.2 Options within the command

V-type
Switches the selected corner reliefs to V-type.

Note: Only corner reliefs which are related to exactly two bends can be switched to v-type.

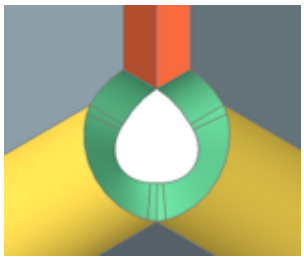


RECtangular
Switches the selected bend reliefs/ corner reliefs to rectangular.

Bend	Corner
	

CIrcular
Switches corner reliefs to circular.

Note: Only corner reliefs which are related to exactly two bends can be switched to circular reliefs.



relief Diameter
Specifies the relief diameter.

Auto
Uses the minimal diameter.

REference point
Specifies the reference point for the relief.

Middle
The reference point is at the intersection of the bend lines.

Corner
The reference point is at the circular arc through the base points of the corresponding rectangular reliefs.

from Global settings
Uses the value from global sheet metal settings of document.

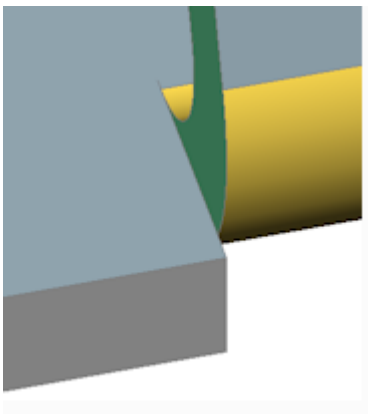
SMooth
Switches the selected bend reliefs to smooth.

Note: Smooth bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



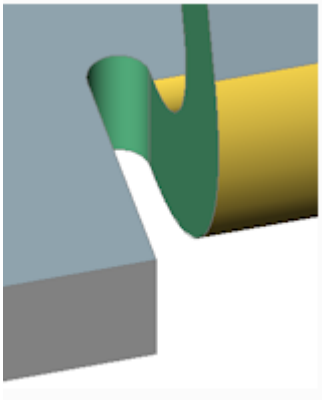
RIp
Switches the selected bend reliefs to rip.

Note: Rip bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



ROund
Switches the selected bend reliefs to round.

Note: Round bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



relief EXtension
Specifies the relief extension.

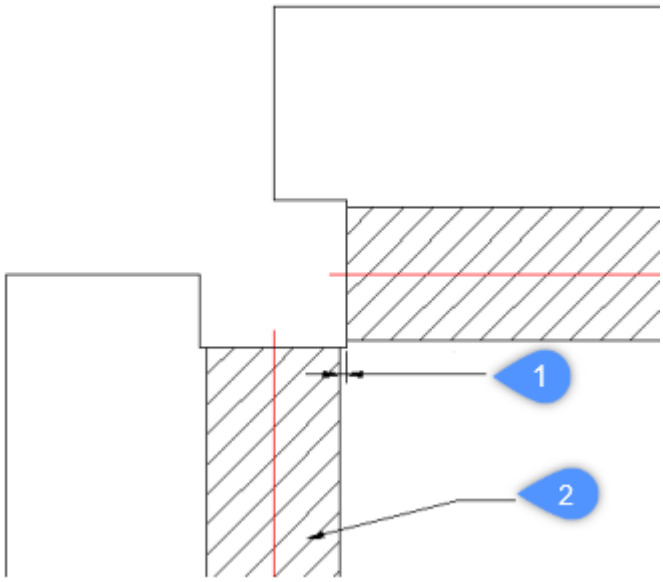
Auto
Keep the current extension.

Entire model
All corner or bend reliefs in all sheet metal parts in the model will be switched to the specified type using the specified parameters.

Note: If this option is chosen as the response to the initial prompt, all corner reliefs switch to V-type.

24.85.3 Relief extension illustration

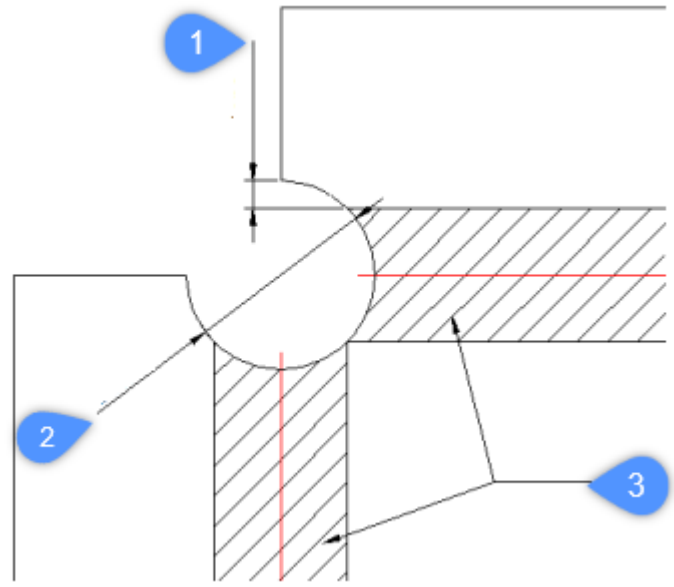
Rectangular and V-type reliefs:



1 Extension

2 Bend

Circular reliefs:



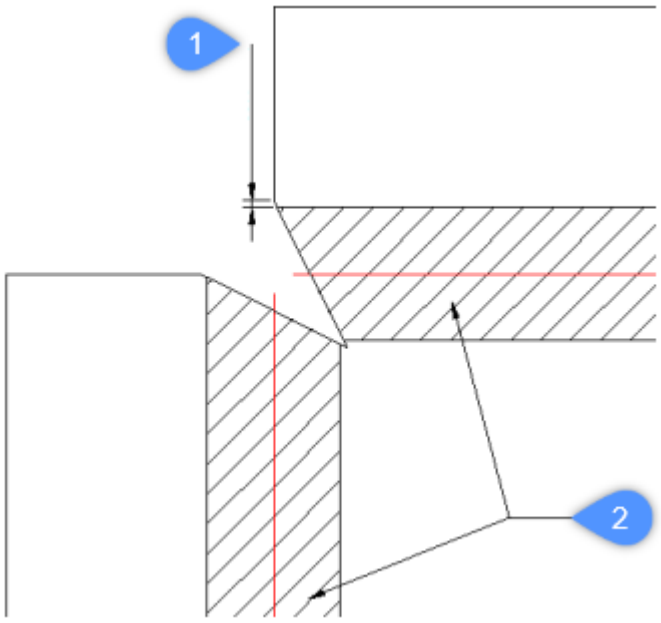
1 Minimal extension

2 Diameter of the relief in the unfolded representation

Note: Reference to determine the center of the relief.

3 Bend

V-type reliefs:



1 Extension

2 Bend

24.86 SMREPAIR command

SmRepair restores the 3D solid model of a sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.86.1 Method

There are three methods:

- Repair by thickening one of the sides

  Note: SmRepair restores the 3D solid model of a sheet metal part by thickening one of its sides. All thickness faces become perpendicular to the flange faces.

- Repair of lofted bend features

  Note: Fixes problems specific to parts with lofted bend features. It merges adjacent bends and provides tangential connections with flanges.

- Repair by converting bends

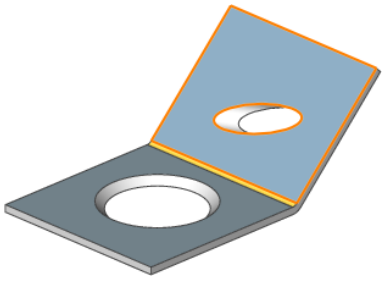
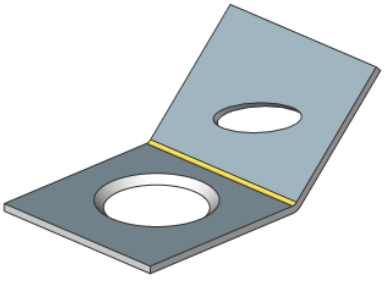
  Note: Converts incorrect bends to regular bends. If a single, incorrect bend is selected, it will be repaired. If a 3D solid is selected, then all incorrect bends will be converted to regular bends.

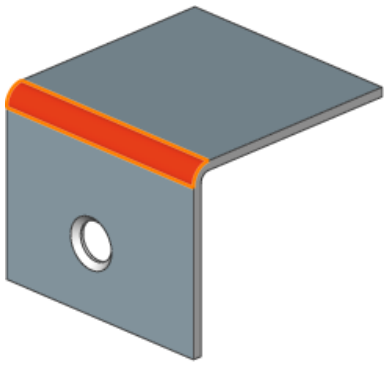
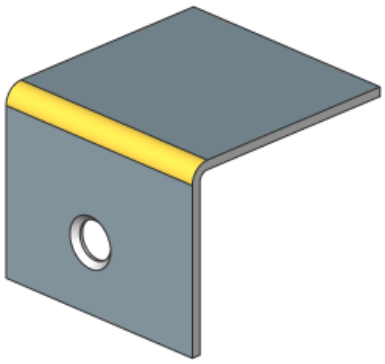
24.86.2 Options within the command

switch on Local repairing

Repairs one selected flange face, lofted bend face, thickness face or side edge. A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created.

Note: The size and position of the opening on the selected face is preserved and the size and position of the opening on the opposite face is adjusted.

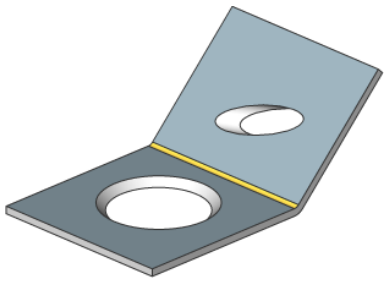
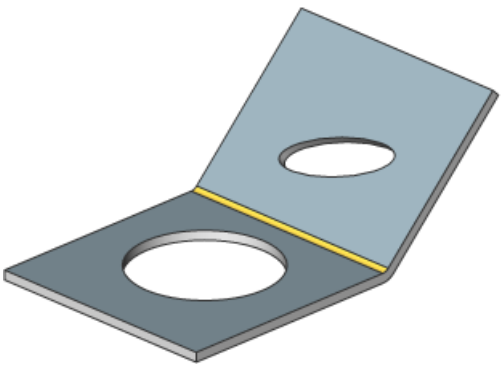
Before (flange face is selected)	After
	

Before (incorrect bend face is selected)	After
	

switch on Global repairing

Repairs all the incorrect flange faces, lofted bend faces, thickness faces or side edges from the selected solid. A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created.

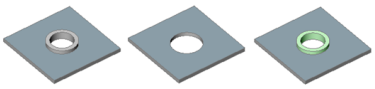
Note: The size and position of the opening on the selected face is preserved and the size and position of the opening on the opposite face is adjusted.

Before	After
	

Scan for lost details

Searches and retains model parts that remain as separated bodies after repair.

Note: The scan tolerance is a fraction of the initial model volume. This fraction will be ignored and only larger bodies will be retained. For example, if the scan tolerance = 0.25, only parts with a volume that is larger than 25% of the initial body will be retained.



From left to right:

- Initial model with an unrecognized form feature.
- Model after SmRepair without scanning for lost details: the circular feature is removed.
- Model after SmRepair with scanning for lost details. Two bodies are created: a horizontal flange with a hole and a separate body (highlighted in green color). The bodies can be unified with the Union command if required.

24.87 SMREPLACE command

Replaces form features in sheet metal parts.

Shape Lite Pro Mechanical BIM

Icon:

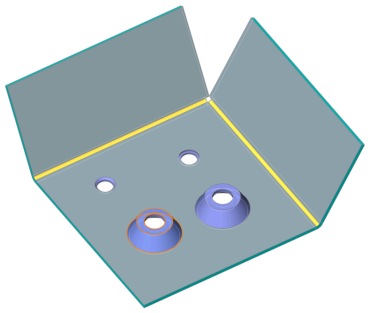
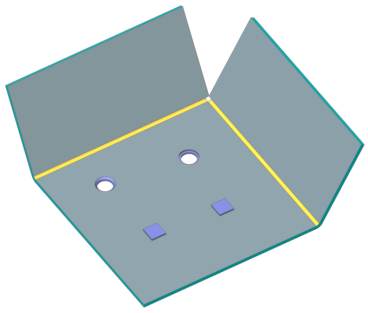
24.87.1 Description

Replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library. Displays the Select file for form feature dialog box where you can select a form feature file.

24.87.2 Options within the command

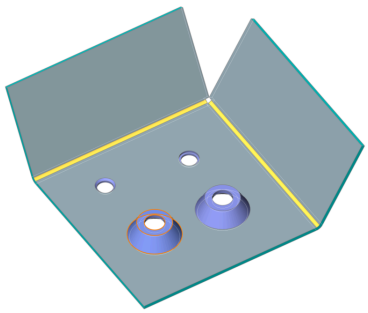
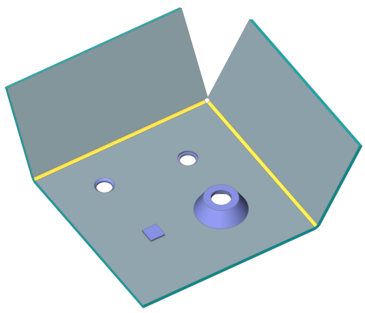
Yes

Replaces all form features of the same type as selected feature(s).

Before	After
	

No

Replaces the selected form feature(s) only.

Before	After
	

parameters Mode
Controls whether the parameter values of the form features that are replaced will be inherited or not. It is defined by the selected mode of replacement.

component Defaults
Parameter values of the new form feature are used.

Inherit from replaced features
If possible, parameter values are copied from the form feature(s) being replaced.

Edit parameters
Allows to edit all parameters.

24.88 SMRIBCREATE command

Creates associative Rib form features on sheet metal.

Shape Lite Pro Mechanical BIM

24.88.1 Description

This command is obsolete. Use the SMBEAD command instead.

24.89 SMROLLEDEDGE command

Creates rolled edges on curved flanges.

Shape Lite Pro Mechanical BIM

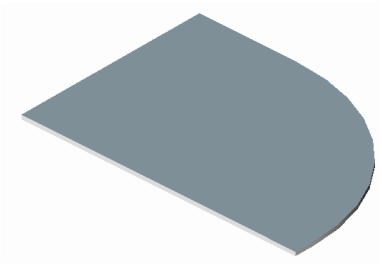
Icon: 

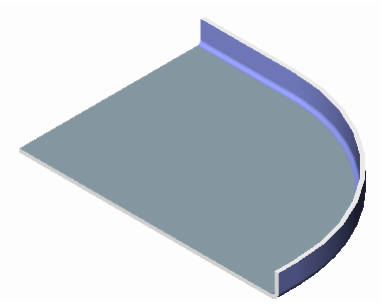
24.89.1 Method

Select the edges on flanges or the flange face (entire flange's side) and specify the length of the rolled edge.

Note: The RolledEdge is not created on edges adjacent to Bend, Jog, BendRelief, CornerRelief.

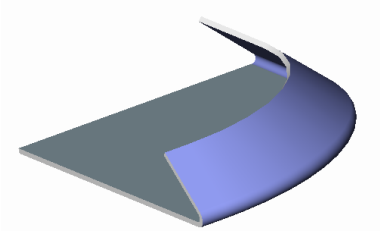
When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.





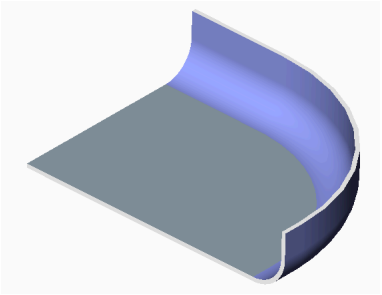
24.89.2 Option within the command

Angle
Specifies the angle of the rolled edge. The default value is 90 degrees.



Note: If the FEATURECOLORS system variable is ON, the rolled edges will be displayed in the color set by SMCOLORROLLEDEDGE system variable.

Radius
Specifies the radius of the rolled edge. The default value is the thickness of the flange.



24.90 SMROTATE command

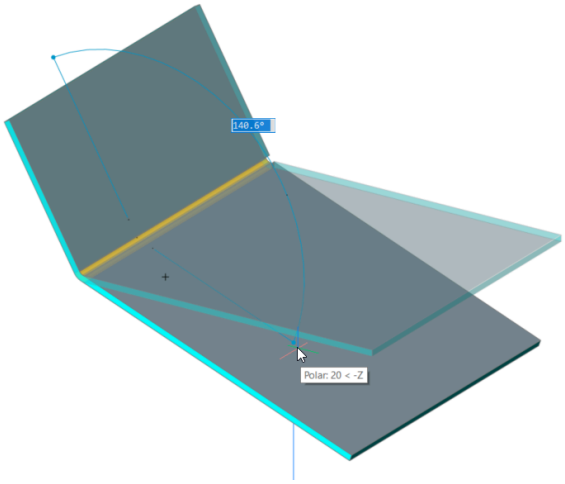
Rotates a selected flange of a sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.90.1 Description

Rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.



Note: A dynamic field angle displays the angle between the selected flange and the base flange. You can press the TAB key to display the absolute rotation angle.

24.91 SMSELECT command

Selects all identical form features in the sheet metal model.

Shape Lite Pro Mechanical BIM

Icons:

24.91.1 Description

Selects hard edges, same form features, non-orthogonal thickness edges, flat edges, and the side of a sheet metal part.

24.91.2 Options within the command

Hard edges
Selects all hard edges in the model.

Same form features
Selects all form features identical to the selected feature.

SImilar form features
Selects all form features similar (the same type, but with different parameter values) to the selected feature.

Non-orthogonal thickness faces
Selects all edges of a specified flange common with non-orthogonal thickness faces.

SIDe of sheet metal part
Selects faces on the same side of a sheet metal part within a selected face.

Note: Requires flange, regular or lofted bend face to start.

Flat edges
Selects all flat edges on a specified solid.

24.92 SMSPLIT command

Splits a flange.

Shape Lite Pro Mechanical BIM

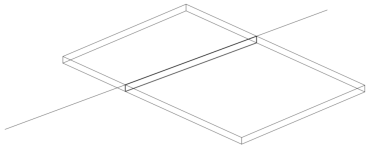
Icons: 

24.92.1 Description

Splits a flange along a polyline drawn along a face. It splits lofted bends along the line on the ruled surface. (Short for Sheet Metal Split).

24.92.2 Options within the command

Select lines or edges to split the flange
Splits the flange according to a selected line or edge.



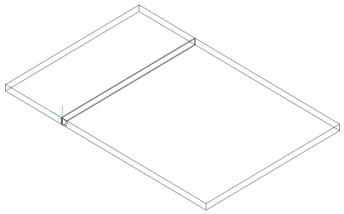
Center
Splits along the center of the polyline.

Left
Splits to the left side of the polyline.

Right
Splits to the right side of the polyline.

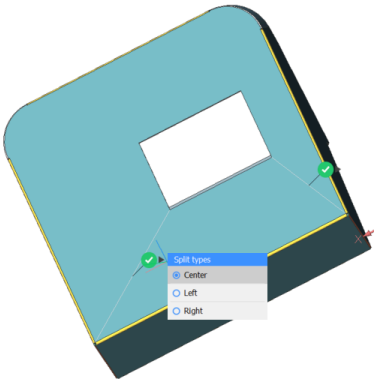
Accept
Accepts the suggested way to split.

SMart split
Splits the flange at a specified point.

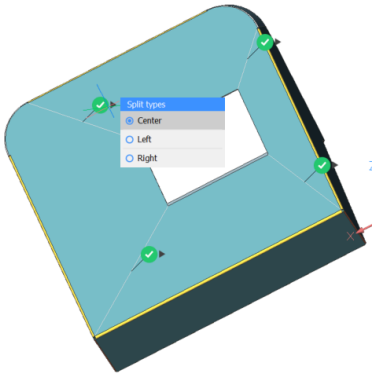


Note: Make sure the 3D entity snaps are toggled on to be able to pick points on the flange.

Flange propagate
Detects all possible splits for the selected flange.



vertex Propagate
Detects all vertices on the flange and applies splits.



Note: The multiple splits suggestions are marked as follows:

 ***: Selected suggestion. Click to switch to not selected.***

 ***: Not selected suggestion. Click to switch to selected.***

draw a New line
Gives the option to draw a line on a selected flange to split the flange.

Note: Make sure the 3D entity snaps are toggled on to be able to pick points on the flange.

24.93 SMTAB command

Creates tab features on sheet metal parts.

Shape Lite Pro Mechanical BIM

Icons: 

24.93.1 Method

There are two main workflows:

- Convert a junction into tabbed junction (in either Single tab or Array tab mode).
- Convert a curved hard edge into a curved tabbed connection of a flange and a bend (or lofted bend).

24.93.2 Option within the command

Single tab

Creates a single tab.

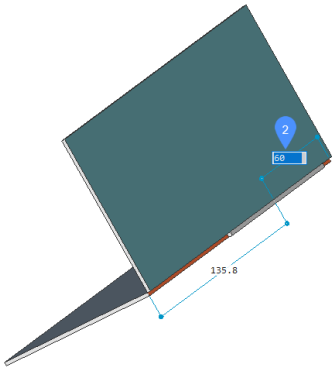
Length

Defines the length of the tab (1). The tab is placed symmetrically with respect to the center point.



Central point

Positions the tab along the junction. The dynamic dimension fields show the distance to the end points of the junction (2). Press the TAB key to switch between the dynamic fields.



Array tab
Creates multiple tabs.

Flip tab
Allows to toggle between two possible configurations when the tab feature is not symmetrical with respect to the junction sides.

slot Number
Defines the number of slots.

Distance
Defines the distance between the tabs.

Length
Defines the length of a tab.

Round radius
Creates a filleted tab with a given fillet radius.

CHamfer distance
Creates a chamfered tab with a given chamfer parameter.

24.94 SMTABCREATE command

Creates tab features on sheet metal parts.

Shape Lite Pro Mechanical BIM

24.94.1 Description

This command is obsolete. Use the SMTAB command instead.

24.95 SMUNFOLD command

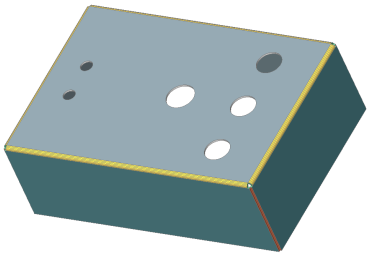
Creates an unfolded 2D or 3D representation of a sheet metal part.

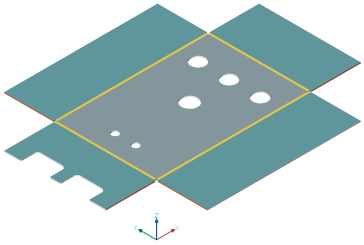
Shape Lite Pro Mechanical BIM

Icons: 

24.95.1 Method

Select the base flange of the sheet metal part, specify a point in the drawing to place the unfolded body and press Enter to keep the unfolded body in the drawing or choose an option.





24.95.2 Option within the command

Associative
Opens an additional window with the model's unfolded representation. Both folded (left) and unfolded (right) representations have bidirectional associativity which allows transferring modifications to the neighbor model automatically when changing window focus.

Note: Tangent bends, jogs (with and without flange) and hems (all types, except closed hems) are also supported.

save 2D geometry
Saves the 2D geometry of the unfolded metal part as a separate drawing file.
From the **Generic Save File** dialog box you have to choose to save the drawing either in Drawing Exchange Format (*.dxf) or in Standard Drawing File (*.dwg).

save 3D geometry
Saves the 3D solid of the unfolded metal part as a separate drawing file.
From the **Generic Save File** dialog box displayed you have to choose to save the drawing either in Drawing Exchange Format (*.dxf) or Standard Drawing File (*.dwg).

Place view to layout
Places the view in a new or an existing layout.

Optimize bend annotations
Optimizes the bend annotations in the exported geometry.

Keep
Places the unfolded 3D solid in the current drawing.

24.96 SNAP command

Sets the cursor snapping properties.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SN

Sets the cursor snapping increments either by specifying the x and y values or between two points, the rotation and the style.

24.96.1 Method

There are three methods to set snapping increments:

- By specifying a single value for both the X and Y axis.

- By specifying a distance between two points as a value for both the X and Y axis.
- By specifying the spacing independently for the X and Y axis.

Note: Independent X and Y spacing work only in the Standard snap style.

24.96.2 Options within the command

snap ON/OFf
Turns on/off snap.

Note: Snap must be turned on in each viewport separately.

Note: The snap can be controlled by SNAPTYPE, POLARDIST, ADAPTIVEGRIDSTEPSIZE and SNAPSTYL system variables.

Rotate
Specify the base point and rotation angle for the snap grid.

snap Style
Toggle between Standard and Isometric snap distances.

Standard
Rectangular snap distance.

Aspect
Specifies the grid spacing independently for the horizontal(X) and vertical(Y) directions.

Isometric
Sets the grid, snap distance and crosshair to the current isometric plane.

24.97 SOLID command

Draws solid-filled polygons.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PLANE, SO

Draws 3 and 4-sided solid-filled regular and irregular 2D polygons.

Note: Does not draw 3D solids.

24.97.1 Method

There are four methods to draw solid-filled polygons:

- By picking points
- ***Note:*** After picking the first point, BricsCAD draws a ghost line to the next point. After specifying the second corner of the polygon, its shape and size can be previewed. If a fourth point is specified, an adjacent polygon is created (the third and fourth points of the first polygon become the first and second points of the second polygon).
- Rectangular
- Square

- Triangle

24.97.2 Options within the command

Rectangle
Draws one or more connected solid-filled rectangles.

Square
Draws one or more connected solid-filled squares.

Triangle
Draws one or more connected equilateral solid-filled triangles.

Note: 2D solids can be edited directly through grips.

24.98 SOLIDEDIT command

Edits 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icons: 

24.98.1 Description

Edits the faces, edges, and bodies of 3D solids and 2D regions.

24.98.2 Options within the command

Face
Edits a face.

Extrude
Extrudes the selected face or region.

Specify height of extrusion
Allows you to specify a distance.

Note: Positive distance extrudes outwards from the face (creates a post). Negative distance extrudes into the body (creates a hole).

Path
Specifies the entity for defining the extrusion shape and direction.

Specify angle of taper for extrusion
Allows you to specify an angle.

Note: Positive angle tapers inwards (get narrower). Negative angle tapers outwards (gets wider).

Move
Moves the selected face.

Vector
Specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point.

Rotate
Slants the selected face about an axis.

Specify the rotation angle
Allows you to specify an angle.

Note: Positive angle rotates the face counterclockwise around the axis. Negative angle rotates the face clockwise.

Offset
Extends the selected face.

Specify offset distance
Allows you to specify a distance.

Note: When entering a positive distance, the face is offset outwards (3D body becomes larger). When entering a negative distance, the face is offset inwards (3D body becomes smaller).

Taper
Tapers (slants) the selected face.

Delete
Deletes the selected face and then repairs the model.

Note: The selected faces are deleted and BricsCAD repairs the 3D solid to keep it solid. This option cannot remove innate faces.

Copy
Copies faces. Selected faces turn into regions.

coLor
Changes the color of the selected face.

Edge
Edits an edge.

Copy
Copies the selected edge and turns it into a line.

Note: The copies edges are turned into entities such as lines and splines.

coLor
Changes the color of the selected edge.

Body
Edits the body of a 3D solid.

Imprint
Imprints 2D onto 3D solids. The 2D entity must lie on, or intersect, one of the 3D solid's faces.

Note: The imprint can be extruded.

seParate
Separates disjointed 3D solids.

Note: This option does not work on bodies created through Boolean operations (Union, Intersect, Subtract).

Shell
Turns the 3D solid into a hollow entity (shelled).

Enter shell offset distance
Allows you to specify a distance.

Note: If a positive distance is entered, the body shells outwards. If a negative distance is entered, the body shells inwards.

Note: If the shell offset distance is larger than 3D solid, the command fails.

cLean
Cleans the 3D solid.

Note: The cleaning operation checks for errors in the 3D solid.

Check
Reports on the 3D solid.

Undo
Undoes the last editing operation.

eXit
Returns to the previous prompt.

24.99 SOLIDIFY command

Converts meshes to solids.

Shape Lite Pro Mechanical BIM

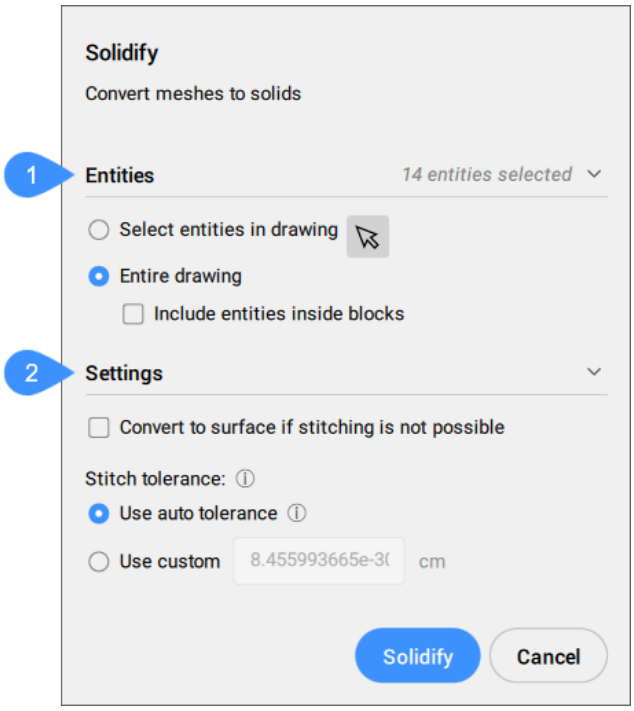
Icon:

24.99.1 Description

Converts regions and (groups of) meshes to solids (and planar surfaces), both in model space and inside block definitions.

The command tries to convert meshes to surfaces, then to stitch surfaces where possible and finally to convert the stitched surfaces to solids. It then tries to remap the properties of each surface or block reference converted (to reapply colors, materials, etc.).

24.99.2 Options within the command context panel



1 Entities

2 Settings

Entities

Select entities in the drawing
Allows you to choose which entities to select.

Entire drawing
By default, the panel selects all entities in the model space.

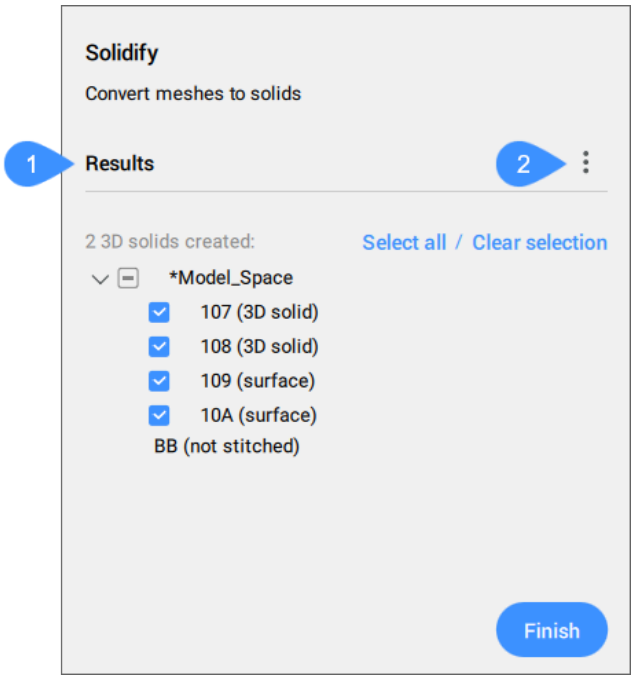
Settings

Convert to surface if stitching is not possible
Tick the option to have the algorithm detect and convert meshes to surfaces.

Stitch tolerance
Defines the maximum gap within which stitching will be performed.

Use auto tolerance
The algorithm automatically determines a stitching tolerance. This option is highly recommended.

Use custom
Allows you to type in a stitching tolerance value expressed in drawing units.



1 Results

2 Result types menu

Results

The command context panel displays the results suggestions in a tree view.

Click the entities names in the list to preview/highlight them in the model space (if possible). To add or remove entities to the preview, use the **Shift** and **Ctrl** keys and the **Select all** and **Clear selection** options.

Right-click an entity name in the list to display a context menu. Use the **Zoom** option to extend the view to the highlighted entities.

Tick the check-boxes to edit the list of entities to be converted.

Press the **Finish** button to accept the selection.

Result types menu

Displays the list of result types (**3D solid**, **surface**, **not stitched**, **could not create surface**).

Select/deselect result types in the menu to filter the displayed results in the tree view.

Note:

- To obtain a cleaner model, you can further use the PURGE, SIMPLIFY, and DMSIMPLIFY/ DMSIMPLIFYALL commands (for BIM license level use also the BIMIFY command).
- The SOLIDIFY command is integrated in the DWGHEALTH command's **Repair 3D Imports** routine. You can also add it to your own custom DWGHEALTH routine (see **DWGHEALTH command** article).

24.100 SOLPROF command

Creates hidden line representations of 3D solids in layout viewports.

Shape Lite Pro Mechanical BIM

Icon:

24.100.1 Method

Select one or more 3D solids and create hidden line representations of them. The representations are blocks.

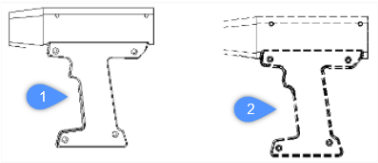
Note: This command operates only in model space of a layout tab viewport. Use the MSPACE command to activate the model space viewport.

Note: This command does not work with any other kind of 3D entity or with 3D solids referenced externally (xrefs). BricsCAD can filter out the ineligible entities.

24.100.2 Options within the command

Use separate layers for hidden lines?
Determines id hidden lines are placed on a separate layer from the visible lines.

Yes
Creates two 2D blocks, one for the visible lines placed on layer PV (1) and another block for hidden lines on layer PH (2).



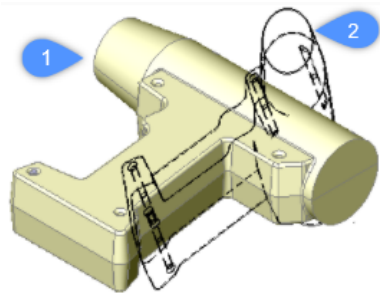
No
Creates one block of both hidden and visible lines on layer PV.

Note: The blocks and layers are named by the program, such as block *U3 and layers PH-88 and PV_88:

- *U identifies blocks created by the program
- PH is short for “profile hidden”
- PV is short for “profile visible”

Project result onto plan?
Determines if the profiles are projected (if the resulting blocks are 2D).

Yes
Profile lines are projected onto the drawing. The 3D solids (1) are projected onto a plane parallel with the layout’s viewport, creating a block in 2D (2).



Note: In this image, the viewpoint has been rotated to show that the block(s) are indeed 2D.

No
No projection, the blocks are made of 3D entities.

Hide tangential edges?
Toggles the display of tangential lines.

Note: Tangential lines are used to show the transition between two faces that are curved. In a 3D shaded model, they are not needed as highlights and shadows indicate the transition. In 2D wireframe mode it can be useful to show them.

24.101 SPELL command

Opens the **Spell checking** dialog box.

Shape Lite Pro Mechanical BIM

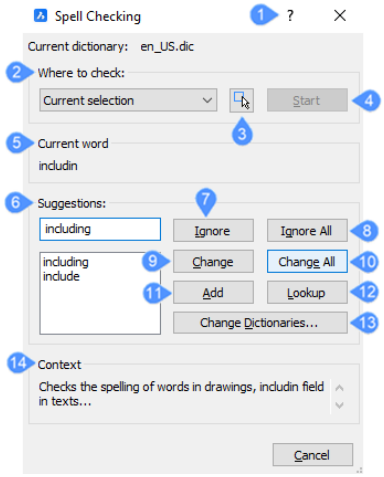
Icon: ABC

Alias: SP

24.101.1 Description

Opens the **Spell checking** dialog box to find and fix potential spelling errors in the current drawing.

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

24.101.2 Command reference

Opens the Bricsys Help article about the SPELL command.

24.101.3 Where to check

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.

24.101.4 Select entities

Select the text to be checked. Choose one or more pieces of text and press Enter to return to the dialog box.

24.101.5 Start

Starts checking the spelling of words.

24.101.6 Current word

Displays the misspelled word.

24.101.7 Suggestions

Lists the suggested replacements for misspelled words. Choose a replacement word and click **Change** or **Change All**.

24.101.8 Ignore

Ignores the spelling of the word.

24.101.9 Ignore All

Ignores all instances of the word.

24.101.1 Change 0

Changes the word to the one selected from the **Suggestion** list.

24.101.1 Change All 1

Changes all instances of the word to the one selected from the **Suggestion** list.

24.101.1 Add
2

Adds the word to the word list.

24.101.1 Lookup
3

Provides additional alternative spellings for the word selected in the **Suggestions** list.

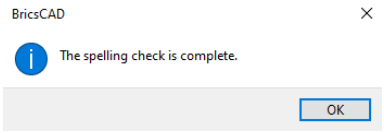
24.101.1 Change Dictionaries...
4

Changes the dictionary used for spell checking. Displays the **Change Dictionaries** dialog box. For more information, see the article **Change Dictionaries dialog box**.

24.101.1 Context
5

Displays the context in which the misspelled word appears.

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



24.102 SPHERE command

Creates a 3D Solid in the shape of a sphere.

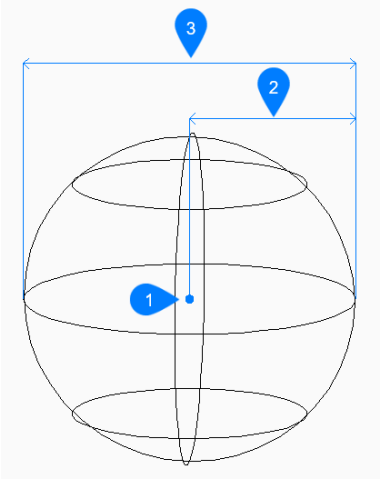
Shape Lite Pro Mechanical BIM

Icon:

Note: In BricsCAD Lite, which doesn't support 3D Solids, the SPHERE command launches the AI_SPHERE command.

24.102.1 Description

Creates a 3D Solid in the shape of a sphere. Choose from a combination of options including center, radius and diameter.



1 Center

2 Radius

3 Diameter

24.102.2 Method

This command has 1 method to begin creating a sphere:

- Center of sphere

Center of sphere
Begin creating a sphere by specifying the center then:

Set radius of sphere
Specify the radius of the sphere.
Additional option: [Diameter]

24.102.3 Option within the command

After you begin creating a sphere, the following option may be available:

Diameter
Specify the diameter of the sphere.

24.103 SPLINE command

Creates a spline.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SPL

24.103.1 Description

Creates a 2D or 3D spline that is open or closed, or converts an object to a spline.

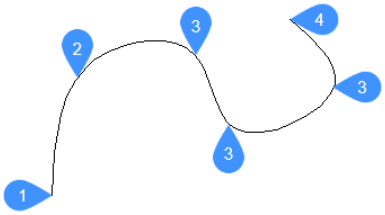
24.103.2 Method

Set the first point, the second point, and the next ones. You can add unlimited points until you press Enter to end the command.

Set first point
Begin creating a spline by specifying the start point.

Set second point
Specify the first fit point.

Set next point
Specify the next fit point.



1 First point of spline (no tangency)

2 Second point

3 Next point

4 End point

24.103.3 Options within the command

Object
Converts selected entities in splines.

start Tangency
Allows you to specify a tangency for the starting point.
Press Enter to ignore this option.

end Tangency
Allows you to specify a tangency for the ending point.
Press Enter to ignore this option and finish the command.

toLerance
Sets the fit tolerance. Specify (in drawing units) how closely the spline matches its fit points. If the fit tolerance is 0, the spline passes through its fit points.

Undo
Removes the last fit point and continues drawing from the previous fit point.

Close
Automatically draws a spline segment between the start and end points to create a closed spline.

24.104 SPLINEDIT command

Modifies the properties of a spline. Converts a spline to a polyline.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SPE

24.104.1 Description

Allows the modification of the fit data (fit points, fit tolerance, tangents at the endpoints and knot parametrization), vertices, the polynomial degree of the spline and the weights of each control vertex. Also allows the joining of the spline with an open 2D entity and reversing the direction of the spline.

24.104.2 Options within the command

Close/Open
Opens or closes the spline by adding or removing a segment between the start and endpoint.

Join
Joins the spline with an open 2D entity that shares an (coincident) endpoint with the spline.

Fit Data
Modifies the fit point data.

Add
Adds a fit point between two existing ones.

Delete
Removes fit points.

Kink
Adds a knot and fit point at the specified location on the spline which does not maintain tangent or curvature continuity at that point.

Move
Moves a fit point.

Purge
Replaces the fit data from the spline with control vertices.

Tangents
Specifies the start and end tangents.

toLerance
Refits the selected spline to the existing fit points using a new tolerance value.

Edit vertex
Edits control frame data.

Add
Adds a new control vertex at the specified point.

Delete
Removes a control vertex.

Elevate order
Increases the polynomial order of the spline (degree plus one) to increase the number of control vertices across the spline. Maximum value is 26.

Move
Relocates control vertices one by one.

Weight
Changes the weight of a specified control vertex then recalculates the spline based on the new value. Larger weights pull the spline closer to the control vertex.

convert to Polyline
Converts the spline to a polyline. The precision determines how closely the resulting polyline matches the spline.

Note: The PLINECONVERTMODE variable specifies the kind of polyline to convert to (0 – with line segments, 1 – with arc segments).

Reverse
Reverses the direction of the spline; a triangle icon appears on what was formerly the spline's endpoint.

Undo
Undoes the previous modification.

eXit
Concludes the editing of the spline or returns to the previous prompt.

24.105 SPOTLIGHT command

Creates a spotlight.

Shape Lite Pro Mechanical BIM

Icon: 

24.105.1 Description

Creates a spotlight, which casts a narrow beam towards a target point.

Note: The options of this command are influenced by the value of the LIGHTUNITS system variable.

24.105.2 Options within the command

Specify source position
Specifies the location of the light in 3D space.

Specify target position
Specifies the location of the target.

Note: The target is the point at which the spotlight is focused.

Name
Specifies the name for the spotlight.

Intensity factor
Specifies the intensity of the light.

Note: Enter an intensity value between 0.00 and the maximum value supported by your system (max float = maximum value of a floating-point number).

Note: When the LIGHTINGUNITS system variable value is 0, this option is named Intensity.

Status
Toggles the use of light.

ON
Light is included when calculating rendering.

OFf
Light is excluded.

Photometry
Specifies the photometric properties of the light.

Note: This option is only available when the LIGHTUNITS value is 1 or 2.

Intensity
Enter the intensity expressed in candela (Cd).

Flux
Enter the luminous flux in lumens (Lm).

Illuminance
Enter the illuminance expressed in lux (Lx) or foot-candles (Fc).

Distance
Enter a distance expressed in drawing units.

Color
Enter a color name.

?
Lists the names of available colors.

Kelvin
Enter a Kelvin temperature expressed in degrees Kelvin.

Hotspot
Specifies the angle of the hotspot, the area of light concentration.

Note: The Hotspot angle represents the angle of the central light cone.

Note: The angle determines the size of the hotspot, regardless the distance of target.

Falloff
Specifies the angle of weaker illuminations surrounding the hotspot.

Note: The Falloff angle represents the angle of the full light cone.

Note: Outside of the falloff, there is no light.

Note: When a Falloff angle is entered that is smaller than the Hotspot angle, the latter is set equal to the Falloff angle. When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot) that is surrounded by an area of lesser intensity (falloff).

Note: 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 the light cone is rather sharp. The greater the difference between both angles, the softer the edge of the light cone.

shadoW
Specifies the look of shadows cast by this light.

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges.

Note: Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources.

Shape

Disk
Defines a circular shadow shape that behaves as an area light.

Rect
Defines a rectangular shadow shape that behaves as an area light.

sAmples
Specifies the sampling size of the shadow.

Note: Larger numbers are more accurate but take longer to render.

Visible
Casts a shadow that represents shapes (more accurate) or that is rectangular (faster to render).

Attenuation
Specifies how the illumination falls off with distance from the light.

attenuation Type
Specifies the type of attenuation.

None
No attenuation so that the distance to the light source has no influence.

Inverse linear
Attenuation is the inverse of the linear distance from the light.

Note: At a distance of 2 units from the light source, the light is half as strong. At a distance of 4 units, the light is one quarter as strong.

inverse Squared
Attenuation is the inverse of the square of the distance from the light.

Note: At a distance of 2 units, the light is one quarter as strong. At a distance of 4 units, the light is one sixteenth as strong.

Use limits
Toggles whether the extent of the illumination is limited.

attenuation start Limit
Defines the point where the light starts to shine, measured from the center of the light.

attenuation End limit
Defines the point where the light stops to shine, measured from the center of the light.

filterColor
Specifies the color of the light.

Note: When the LIGHTUNITS system variable's value is 0, this option is named Color.

Enter true color
The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values).

Index color
Defines the color name by the index color.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness.

Colorbook
Enter a name from a colorbook.

eXit
Concludes the command and creates the light.

24.106 SSX command (Express Tools)

Creates a selection set based on a selected entity using filters.

Shape Lite Pro Mechanical BIM

24.106.1 Method

Creates a selection set that contains entities similar to or exactly the same as the selected entity.

Note: To access the selection set for a subsequent command, use the **Previous selection** option.

24.106.2 Options within the command

Block name
Allows you to specify a block name to add or remove from the selection filter.

Color
Allows you to specify a color number to add to or remove from the selection filter.

Entity
Allows you to specify an entity to add to or remove from the selection filter.

Flag
Turns the Entities follow flag On or Off.

Layer
Allows you to specify a layer name to add to or remove from the selection filter.

Ltype
Allows you to specify a linetype name to add to or remove from the selection filter.

Pick
Allows you to select an entity.

Style
Allows you to specify a text style name to add to or remove from the selection filter.

Thickness
Allows you to specify a thickness value to add to or remove from the selection filter.

Vector
Allows you to specify an extrusion vector to add to or remove from the selection filter.

24.107 STANDARDPARTSPANELCLOSE command

This command is obsolete.

Shape Lite Pro Mechanical BIM

24.107.1 Description

Use LIBRARYPANELCLOSE instead.

24.108 STANDARDPARTSPANELOPEN command

This command is obsolete.

Shape Lite Pro Mechanical BIM

24.108.1 Description

Use LIBRARYPANELOPEN instead.

24.109 STANDARDS command

Adds and edits CAD standards for the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

24.109.1 Method

This command associates one or more standard (DWS) files to the current drawing.

A standards file defines common properties for many drawing files to maintain consistency throughout those files. This consistency might be important in collaborative environments, where many people are involved in creating sets of drawings that should comply with some predefined rules.

Standards are created for the following named objects:

- Layers

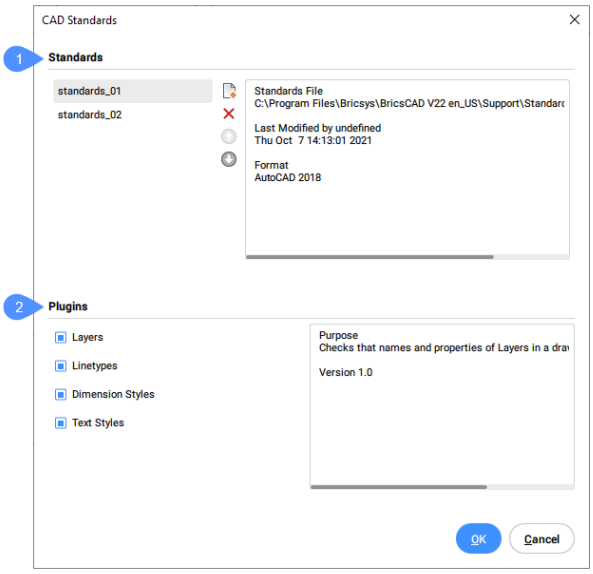
- Text styles
- Line types
- Dimension styles

Standard (DWS) files can be created by the SAVEAS command, when you choose Standards File (*.dws) from the Save as type list.

This command has an associated dialog box that manages all the operations associated with this command.

24.109.2 STANDARDS dialog box

The **CAD Standards** dialog box allows to select the options for the STANDARDS command.



1 Standards

2 Plugins

Standards

This section is split into two sides.

The left side contains a list with the names of the standards already loaded into the drawing and a set of buttons for managing the list.

The names of the standards are the same as the names of the files selected as standards.

The buttons for managing the list are from top to bottom:

- **Adds Standards file**: adds the selected files to the list of standards.
- **Removes Standards file**: removes the selected file from the list of standards.
- **Moves Standards file one position up**: moves the name selected in the list one position up.
- **Moves Standards file one position down**: moves the name selected in the list one position down.

The right side of the **Standards** section displays some properties of the standard selected in the list. These properties include the path of the file containing the standard, the date of creation and the file format.

Standard (DWS) files can be created by the SAVEAS command, when you choose Standards File (*.dws) from the Save as type list.

Plugins
This section contains a list of plug-ins to be used by a checking operation.
A plug-in is a property or a set of properties of the entities in the standards files. When the corresponding checkbox of a plug-in is checked, the checking operation will include that plug-in.
The available plug-ins are as the following:

Layers
The layers defined in the standard files.

Linetypes
The line types defined in the standard files.

Dimension Styles
The dimension styles defined in the standard files.

Text Styles
The text styles defined in the standard files.

24.110 STATBAR command

Toggles the STATUSBAR system variable.

Shape Lite Pro Mechanical BIM

24.110.1 Description

Toggles the STATUSBAR system variable to show or hide the status bar. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'STATBAR.

- On: turns on the STATUSBAR system variable.
- Off: turns off the STATUSBAR system variable.
- Toggle: changes the STATUSBAR system variable to the opposite of the current setting.

24.111 STATUS command

Reports the status of the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

24.111.1 Description

Displays the status report of the drawing in the Command line.

Note: To see the full report press F2 key.

24.112 STLOUT command

Export 3D solids and watertight meshes.

Shape Lite Pro Mechanical BIM

Icon:



24.112.1 Description

Exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing).

24.112.2 Method

There are two methods to use STLOUT command:

- Export 3D solids.
- Export meshes.

24.112.3 Options within the command

Create binary STL file
Determines whether the output file is in binary or ASCII format.

Note: Binary STL file are more compact, while an ASCII text STL file is human-readable.

Select smoothness of objects exported to STL file
Adjusts the quality of the exported entities.

Note: When exporting 3D solids to STL, curved faces become faceted. Use the FACETRES variable to determine the size of the facets:

- Minimum: 0.01 for coarser curves but smaller file size.
- Default: 0.5.
- Maximum: 10 for very fine curves but larger file size.

Low
Exports with FACETRES = 0.5.

Medium
Exports with FACETRES = 2.0.

High
Exports with FACETRES = 10.

Default
The modeler properties from Settings are used.

Note: The modeler properties override FACETRES to control smoothness by using different parameters.

24.113 STOPSCRIPT command

Stops the recording of scripts.

Shape Lite Pro Mechanical BIM

Icon:

24.113.1 Description

Stops the recording of scripts begun by the RECSCRIPT command.

24.114 STRETCH command

Stretches 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: S

Note: This command does not stretch texts, shapes, and 3D solids.

24.114.1 Method

Select an area in your drawing using either a rectangular window or a polygon, then specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance.

24.115 STRING command

Creates a Civil String entity - 3D object made of line and curve segments.

Shape Lite Pro Mechanical BIM

Icon: 

24.115.1 Description

Create a Civil String entity using 3D lines and curves.

Strings are organized in String Groups. Strings within the same group have the option to interact with each other when they cross, creating an elevation point at the intersection. They cross through the exact same elevation.

Note: The Civil Strings can be used as grading input entities, TIN surface breaklines, and corridor baselines. They fill the gap between the (3D) polylines and Alignments functionality.

24.115.2 Method

To create a Civil String, specify the points and their elevations.

Note: Strings can be imported from Survey data, LandXML, or from Civil 3D drawing that contains Feature Lines.

24.115.3 Options within the command

Surface
Allows you to select a TIN surface from which to extract the elevation.

Difference
Allows you to specify an elevation difference from the previous point.

sLope
Allows you to specify a slope (1:x).

Grade
Allows you to specify a grade in %.

Elevation
Allows you to specify an elevation of the point.

Arc
Allows you to draw an arc by specifying its end point.

Close
Automatically draws a 3D line from the end point of the last segment to the start point of the first segment, and ends the command.

24.116 STRINGADDCURVE command

Adds curves to a Civil String.

Shape Lite Pro Mechanical BIM

Icon: 

24.116.1 Description

Adds a curve in a PI point of a Civil String entity with the specified radius value.

Note: The default radius value is 10.

24.116.2 Method

Select the Civil String entity and specify each PI point to add a curve.

24.116.3 Options within the command

Radius
Allows you to set the radius value.

24.117 STRINGADDELEVATIONPOINT command

Adds elevation points to a Civil String entity.

Shape Lite Pro Mechanical BIM

Icon: 

24.117.1 Method

Select the Civil String entity, specify a point and its elevation.

24.117.2 Options within the command

Surface
Allows you to select a TIN surface from which to extract the elevation.

24.118 STRINGADDPI command

Adds PI (Points of intersections) to a Civil String entity.

Shape Lite Pro Mechanical BIM

Icon: 

24.118.1 Description

Adds PI points to a Civil String entity.

24.118.2 Method

Select the Civil String entity, specify each PI point and its elevation.

24.118.3 Options within the command

Surface
Allows you to select a TIN surface from which to extract the elevation.

24.119 STRINGDELETEELEVATIONPOINT command

Deletes elevation points from a Civil String entity.

Shape Lite Pro Mechanical BIM

Icon: 

24.119.1 Method

Select the Civil String entity and click to select the elevation point to delete.

24.120 STRINGDELETEPI command

Deletes PI points (Points of intersections) from a Civil String entity.

Shape Lite Pro Mechanical BIM

Icon: 

24.120.1 Method

Select the Civil String entity and specify each PI point for deleting.

24.121 STRINGEDITOR command

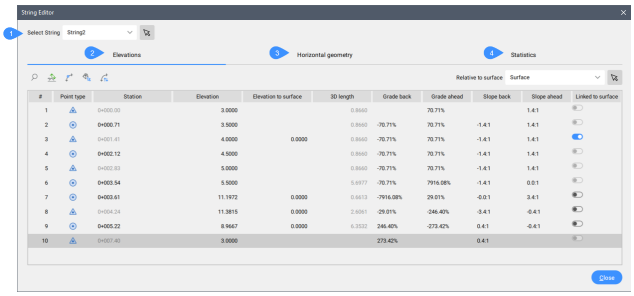
Edits Civil String entities.

Shape Lite Pro Mechanical BIM

Icon: 

24.121.1 Method

Select Civil String entities to open the **String Editor** dialog box.



1 Select String

2 Elevations tab

3 Horizontal geometry tab

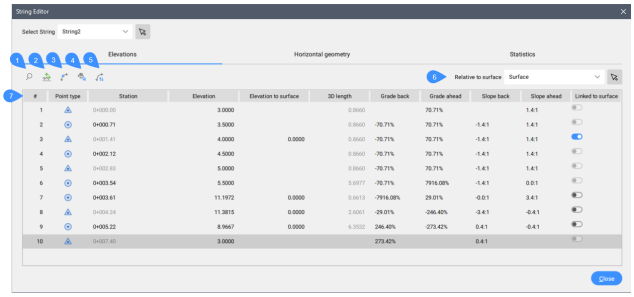
4 Statistics tab

24.121.2 Select String

Displays the Civil String for editing. You can select another Civil String from the drop-down list or click to pick it from the drawing area.

24.121.3 Elevations tab

Lists all points properties of the Civil String for editing.



1 Zoom

2 Raise/Lower

3 Add elevation point

4 Get elevation from surface

5 Reverse string

6 Relative to surface

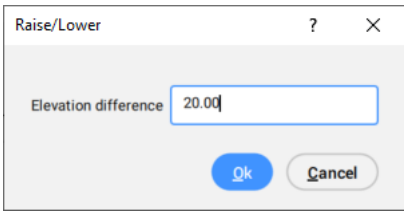
7 String points properties list

Zoom

Zooms in the drawing to the selected Civil String.

Raise / Lower

Opens the **Raise/Lower** dialog to set the elevation difference value. This value will be added to the elevations. Negative values decrease the elevation.



Add elevation point
Launches the STRINGADDELEVATIONPOINT command. Adds an elevation point to a Civil String entity.

Get elevation from surface
Opens the **Elevations From Surface** dialog box.

Reverse string
Changes the direction of the Civil String.

Relative to surface
Displays the selected TIN surface to which a Civil String elevations will be relative. You can select another TIN surface from the drop-down list or click to pick it from the drawing area.

String points properties list
Lists all String points properties.

Note: Right-click a string point properties line to open a context menu with **Zoom to**, **Pan to**, **Copy to clipboard**, and, depending on the point type, the **Delete PI** ( ) or **Delete elevation point** ( ) options.

Position number
Lists the position number of the point in the String.

Point type
Displays the type of point represented by different icons.

- Points intersection (PI)
- Point intersection (PI) read only
- Point intersection (PI) shared
- Elevation Point

Station
Displays the Station value of the point.

Note: The station value for the point intersections (PI) is read-only.

Elevation
Displays the elevation value of the point.

Elevation to surface
Displays the elevation difference between the surface and String point.

3D length
Displays the 3D length of the segment between the current point and the next point.

Grade back
Displays the grade of the current segment from its end to start.

Grade ahead
Defines the end elevation of the current segment in forward direction.

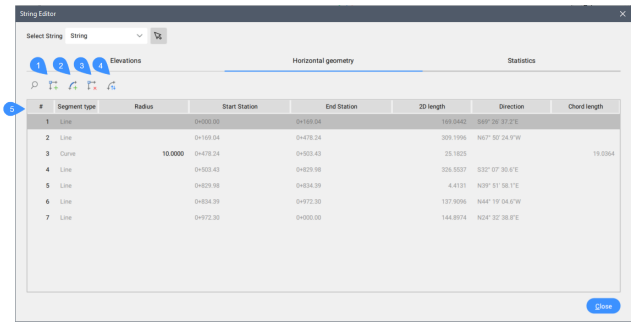
Slope back
Displays the slope of the current segment from its end to start.

Slope ahead
Defines the end elevation of the current segment in forward direction.

Link to surface
If ticked, the String is linked to TIN surface. Civil String elevations change dynamically with TIN surface elevations changes.

24.121.4 Horizontal geometry tab

Lists all segments properties of the Civil String for editing.



1 Add PI

2 Add curve

3 Delete PI

4 Reverse string

5 String segments properties

Add PI
Launches the STRINGADDPI command.

Add curve
Launches the STRINGADDCURVE command.

Delete PI
Launches the STRINGDELETEPI command.

Reverse string
Changes the direction of the Civil String.

String segments properties
Lists all String segments properties.

Position number
Lists the position number of the segment in the String.

Segment type
Displays the segment type.

Radius
Displays the radius value.

Start Station
Displays the start station of the segment.

End Station
Displays the end station of the segment.

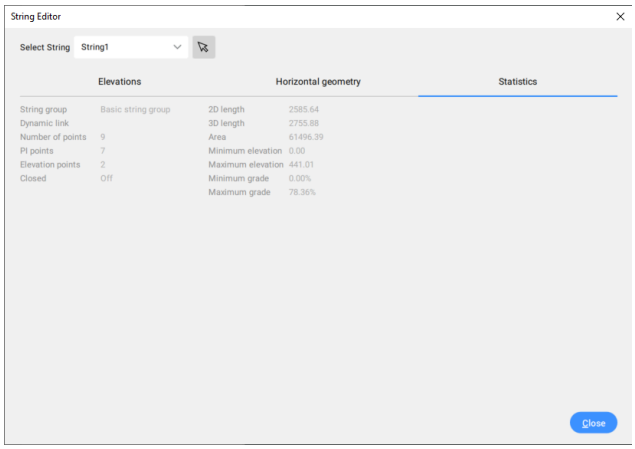
2D length
Displays the 2D length of the segment.

Direction
Displays the direction of the segment.

Chord length
Displays the chord length.

24.121.5 Statistics tab

Lists the general properties of the Civil String.



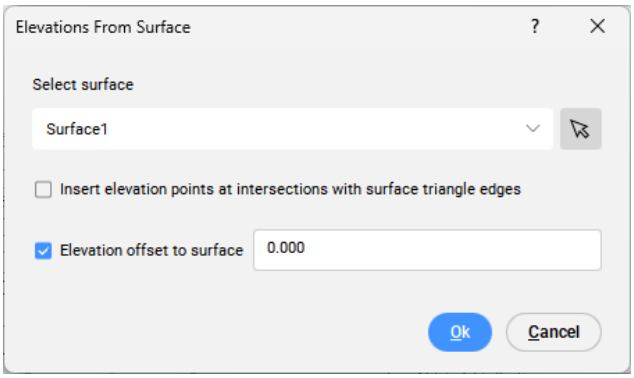
24.122 STRINGELEVATIONFROMSURFACE command

Assigns the elevations of a surface to the geometry and elevation points of a Civil String entity.

Shape Lite Pro Mechanical BIM

24.122.1 Description

Opens the **Elevations From Surface** dialog box to set the elevation parameters from surfaces.



24.122.2 Select surface

Displays the selected surface. You can select a surface from the drop down list or click to pick it from the drawing area.

24.122.3 Insert elevation points at intersections with surface triangle edges

If ticked, inserts the elevation point at each point where Civil String crosses a TIN surface triangle edge.

24.122.4 Elevation offset to surface

If ticked, you are able to define whether Civil String entity is created above or below the TIN surface elevation by specifying a value for an elevation offset to the surface.

24.123 STRINGFROMOBJECT command

Creates a Civil String entity from an existing entity.

Shape Lite Pro Mechanical BIM

Icon:

24.123.1 Description

Create a Civil String entity from an existing entity, such as line, arc, polyline, or 3D polyline.

Note: Civil Strings can be used as grading input entities, TIN surface breaklines, and corridor baselines. They fill the gap between the (3D) polylines and Alignments functionality.

24.123.2 Method

Select the entity (line, arc, polyline, or 3D polyline) that will be converted to a Civil String entity.

24.124 STRUCTUREPANEL command

Opens the **Structure** panel.

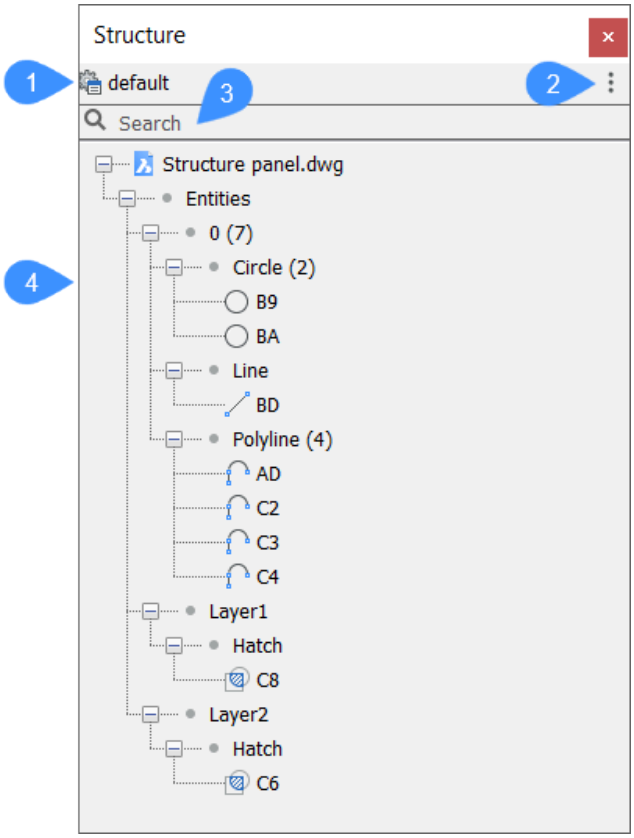
Shape Lite Pro Mechanical BIM

24.124.1 Description

Opens the **Structure** panel to display it in the current workspace. The **Structure** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Structure** panel can be either floating, docked or stacked.

The **Structure** panel enables you to:

- View the entire content of a drawing listed as a structure tree configured by predefined rules.
- Decide how the entities are displayed in the structure tree by selecting a **Structure Tree Configuration** or creating a new one via the **Configure Structure Tree** dialog box.
- Select specific entities from the structure tree to identify them in the model space (and vice versa) and to modify their properties.
- ***Note:*** BricsCAD does not support modifying sub-entities of blocks using the **Structure** panel.



1 Structure tree configuration

2 Configurations menu

3 Search bar

4 Structure tree

24.124.2 Structure tree configuration

Displays the name of the structure tree's active configuration. To edit the active configuration or create a new one, open the **Configure Structure Tree** dialog box by clicking on the configuration's name. Here you can create rules by selecting entity properties as filter, group and sorting criteria (see **Configure Structure Tree dialog box** article).

24.124.3 Configurations menu

This drop-down menu lists the configuration (CST) files stored in the **Support Folder**. Select one to set it as the active Structure Tree Configuration.

default
Displays entities sorted and grouped by layer-based structure.

mechanical
Displays entities sorted and grouped by based structure.

bim
Displays BIM entities sorted and grouped by based structure.

bim - Composition
Displays BIM entities sorted and grouped by composition. For example, you can see/select all concrete walls on all floors at once, or see at a glance which compositions are used.

bim - Profiles
Displays BIM entities sorted and grouped by profile.

bim - Type
Displays BIM entities sorted and grouped by BIM-type. For example, you can see/select all walls on all floors at once.

Configure structure tree
Opens the **Configure Structure Tree** dialog box to edit the existing configurations or to add your own structure tree configurations (see **Configure Structure Tree dialog box** section below).

Select Configuration…
Allows you to load **Structure Tree Configuration** files (*.cst).

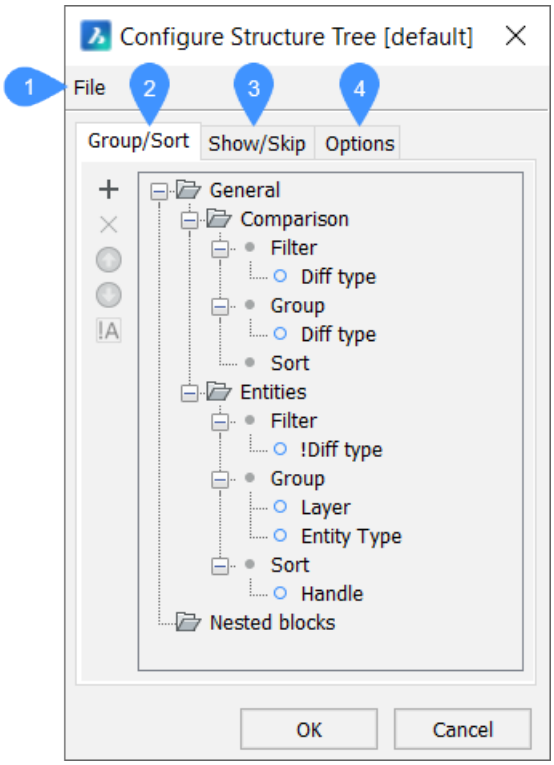
> ***Note:*** The default path were predefined CST files are stored is: *C:\Program Files\Bricsys\V25x64\UserDataCache\Support\en_US*.

24.124.4 Search field

Type in a string to search the structure tree for character strings that match what you typed. The structure tree adapts accordingly, displaying only the result entities.

> ***Note:*** The search field allows the use of case-sensitive query language. For example, type '*$EntityType=Line and (Length>400 and Length<1000)*' to find lines with lengths between 400 and 1000 units.

The **Configure Structure Tree** dialog box allows you to define the **Structure Tree Configuration** of the **Structure** panel.



1 File

2 Group/Sort tab

3 Show/Skip tab

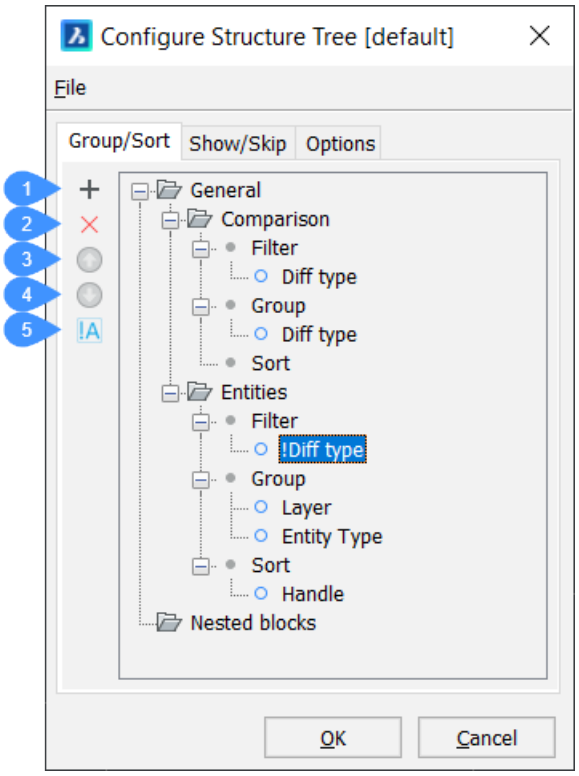
4 Options tab

24.124.5 File

Click the **File** menu in the dialog box and choose one of the available options in the list:

- **Open...**: opens the **Select Structure Tree Configuration File** dialog box to select another CST configuration file.
- **Save**: saves the current configuration.
- **Save As...**: saves the current configuration under a different name.

24.124.6 Group/Sort Tab



1 Add rule or property

2 Delete rule or property

3 Move property up

4 Move property down

5 Toggle negation (filter properties only)

Add rule or property (+)

Adds a new rule or a new property for **Filter**, **Group**, or **Sort**.

Note: The position of the rules and properties is essential, as the rule structure is hierarchical.

- 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 (or at the end of the rule hierarchy when no rule is selected).

  Note: To rename the rule, select it and click its name or right-click and choose **Rename** in the context menu.

- 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 to let you choose a property for filtering, grouping, or sorting.

- Select a property, then double-click or click **OK**.

  Note: Properties and value definitions that are defined with the BIMPROPERTIES command can be selected to configure the structure tree.

- Optionally, type a search string in the **Search** field, then click the **Find previous** (**Shift+F3**) () or **Find next** (**F3**) () arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more filter/ grouping/ sorting properties.

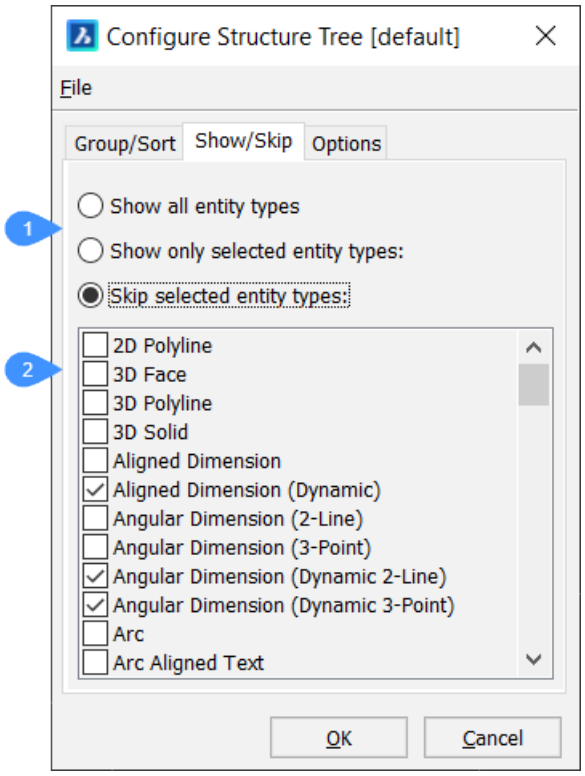
Delete rule or property ()
Deletes the selected rule or property. You can also use the **Delete rule**/**Delete property** option from the context menu of the selected rule/property.

Move property up ()
Moves up the selected rule or property.

Move property down ()
Moves down the selected rule or property.

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

24.124.7 Show/Skip Tab



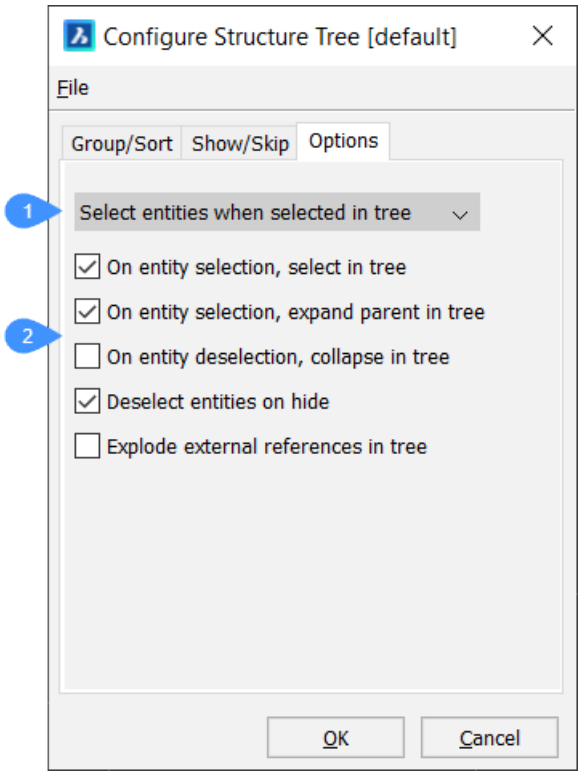
1 Show/skip options

2 Selection list

Show/skip options
Specifies whether selected entities should be shown or not.

Selection list
Allows you to create a selection of entity types to show or exclude.

24.124.8 Options Tab



1 Selection options

2 Additional options

Selection options

Defines what happens in the model when entities are selected in the structure tree. Several options are available:

- **Ignore tree selection**: entities are not selected in the drawing.
- **Highlight entities when selected in tree**: entities in the drawing are highlighted, but not selected.
- **Select entities when selected in tree**: matching entities are selected in the drawing.

Additional options

Defines what happens in the structure tree when an entity is selected/deselected in the model.

On entity selection, select in tree

When on, selects matching entity(ies) in the structure tree when you select them in the drawing.

On entity selection, expand parent in tree

When on, expands the node related to the entity(ies) selected in the drawing.

On entity deselection, collapse in tree

When on, collapses the node related to the entity(ies) selected in the drawing.

Deselect entities on hide

When on, deselects hidden entities.

Explode external references in tree

When on, lists the contents of externally-referenced drawings in the structure tree.

See also the **Working with the Structure panel** guide linked in the **Related information** section which provides a step by step example on how to use the **Structure** panel.

24.125 STRUCTUREPANELCLOSE command

Closes the **Structure** panel.

Shape Lite Pro Mechanical BIM

24.125.1 Description

Closes the Structure panel to hide it from the current workspace. If the **Structure** panel is stacked when you close it, the Structure tab or icon is removed from the stack.

24.126 STYLE command

Opens the **Drawing explorer** dialog box with **Text Styles** selected.

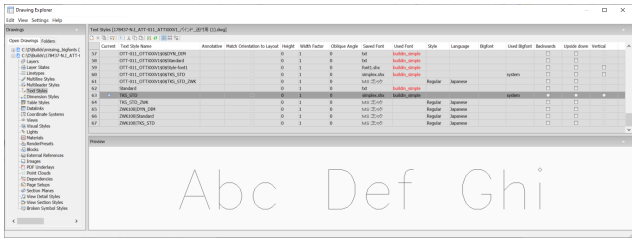
Shape Lite Pro Mechanical BIM

Icon:

Alias: DDSTYLE, EXPFONTS, EXPSTYLE, EXPSTYLES, ST

24.126.1 Description

Opens the **Drawing Explorer** dialog box with **Text Styles** category selected to view and modify styles in the current drawing.



24.126.2 Options within the Drawing Explorer

Current
Designates the current text style. This is the style used by default by the TEXT and MTEXT commands.

Text Style Name
Specifies the style name.

Annotative
Sets the **Annotative** property of the text entities created with this style.

> ***Note:*** When the text style is an annotative one, the **Height** property specifies the height in paper space, while in model space the height depends on the current annotation scale (specified by the CANNOSCALE and CANNOSCALEVALUE variables).

Match Orientation to Layout
Matches the orientation of the text to the orientation of the paperspace layout.

Note: This option is for annotative text only.

Height
Specifies the height of the text. Enter 0 or a number larger than 0:

- 0: does not set the height. The TEXT command prompts for height.
- Not 0: specifies the height of text. The TEXT command does not ask for height.

Width Factor
Specifies the width factor of the text. For example, entering 2 makes the text twice as wide.
Enter a number larger or smaller than 1:

- Less than 1: text is squeezed narrower.
- More than 1: text is spread wider.

Oblique Angle
Specifies the oblique angle for the text. For example, entering 15 slants the text forward by 15 degrees.
Enter a positive or negative number:

- Negative number: text slants to the left.
- 0: text is upright.
- Positive number: text slants to the right.

Saved Font
Specifies the SHX or TTF font file to use. Choose a name from the drop-down list:

- SHX: compiled shape files created for AutoCAD® and BricsCAD.
- TTF: TrueType font files created for Windows and Macintosh.

Note: The program can use any SHX and TTF font found on your computer.

Used Font
Shows the name of the font file found. If the font defined in by **Saved Font** is not found, SHX fonts are replaced by the font defined by the FONTALT (Alternative font) system variable or by substitutes defined with the **Font Substitution** dialog box.

Note: TTF fonts are replaced according to the replacement table defined by the FONTMAP system variable. By default, this is the Default.fmp file stored in the Support subfolder of the roamable root folder (*C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*).

Note: Fonts that have been replaced are displayed in red.

Style
Specifies style for TTF fonts. Choose a style from the drop-down list:

- **Regular**: applies neither bold nor italics
- **Bold**: applies boldface
- **Italic**: applies italics
- **Bold Italic**: applies bold and italics

Note: This property is not available for SHX fonts.

Language
Specifies the language for TTF fonts. Choose one from the drop-down list.

Note: Available languages may vary for each font. This property affects accents and other attributes specific to languages.

Note: This property is not available for SHX fonts. Use **Bigfont** property instead.

Bigfont
Lists the names of SHX bigfont files. Choose one from the drop-down list.

Note: "Bigfonts" is the name given to large SHX files that contain characters for multiple languages.

Note: This property is not available for TTF fonts. Use **Language** property instead.

Backwards
Toggles backwards text. Backwards text is useful when text is plotted on the back of Mylar or other clear media.

Upside down
Toggles upside-down text.

Vertical
Toggles vertical text. Only some SHX fonts support vertical text. This option is not available for TTF fonts.

Preview
Shows what the current style looks like: the font with the properties applied.

24.126.3 Context Menu Options

New
Loads additional Text Style definitions into the drawing.

Delete
Deletes Text Style definitions from the drawing. The current style and the Standard style cannot be deleted.

Purge
Removes unused fonts.

Rename
Renames the selected style.

Select All
Selects all styles.

Invert selection
Deselects the current selection and vice versa.

Set current
Specifies the current working style. Only one style can be current at a time.

Fonts substitution...
Opens the **Font Substitution** dialog box.

24.127 -STYLE command

Creates and modifies text styles.

Shape Lite Pro Mechanical BIM

Alias: FONT

24.127.1 Description

Creates and modifies text styles that set the look of text, such as its font and height. It is meant for macros. Styles determine the look of text in single-line text, mtext, attributes, tables, and dimensions and leaders, but not in dimensional constraints.

24.127.2 Options within the command

?
Lists existing styles and some of their properties including style name, height, generation.

Text style to create or modify
Specifies the style name.

Font type
SHX: font files created for DWG-based programs like BricsCAD.
TTF: font files created for Windows and macOS, and can be used by BricsCAD.

Height
Indicate the height of the text, measured from the baseline to the top of the ascenders.

Width
Makes characters wider or narrower.

Obliquing angle
Slants text to the left for a negative number, to the right for a positive number and keep text upright for value 0.

Draw text backwards
Toggles backwards text.

Draw text upside-down
Toggles upside-down text.

Draw text vertically
Toggles vertical text. This option is available only with certain SHX font files and not at all with TTF font file.

Annotative
An annotatively-scaled style adjusts its height in paper space automatically, according to the plot scale.
You can toggle the creation of an annotative text style, match the text orientation to layout.
When the text style is an annotative one, the text height property specifies the height in paper space, while the annotation scale sets the height in model space.

24.128 STYLESMANAGER command

Creates and edits plot style files.

Shape Lite Pro Mechanical BIM

24.128.1 Method

This command displays a file manager dialog box from which a CTB or STB file can be selected. Double click on a file to display the **Plot Style Table Editor** dialog box or on Create Plot Style Table to create a plot style from scratch or from an existing table.

Note: Plot styles fine-tune how drawings are printed but are not necessary for printing. BricsCAD supports two groups of plot styles to control the printed output:

- CTB color tables simply use colors.
- STB style tables use an extensive range of properties.

24.129 SUBTRACT command

Performs Boolean subtraction operation.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SU

Note: In the BricsCAD Lite license level, the command applies to region entities only.

24.129.1 Description

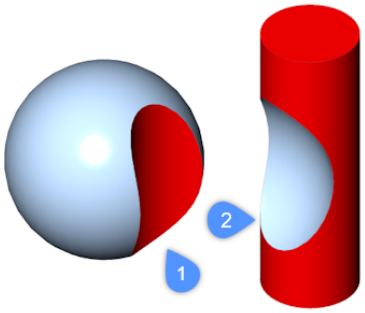
Performs Boolean subtraction operations on 3D solids and 2D regions by removing the overlapping portions of one from another.

24.129.2 Method

Select the object to subtract from and the objects to subtract. The DELETETOOL system variable determines whether the source entities are deleted or retained.

Note: The result of this command is sensitive to the order in which you select entities.

- A cylinder was subtracted from a sphere (1) and vice versa (2).



- The order in which you select entities also applies to regions.

Note: You can interrupt the command by pressing the CANCEL button.

24.130 SUNPROPERTIES command

Opens the **Drawing Explorer** dialog box with **Lights** selected.

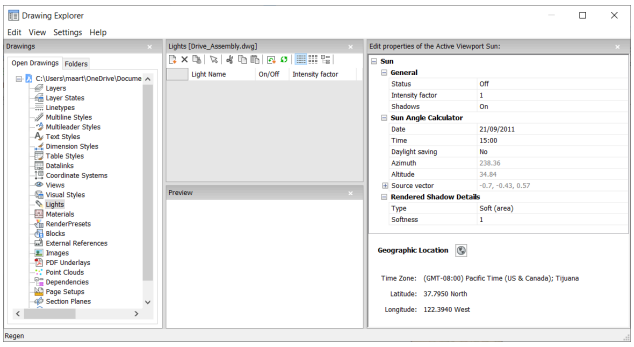
Shape Lite Pro Mechanical BIM

Icon:

Alias: SUN

24.130.1 Description

Opens the **Drawing Explorer** with **Lights** category selected to view and modify sun properties in the current drawing.



24.130.2 Options within the Drawing Explorer

General

Status

Toggles the sun on and off.

Intensity factor

Specifies the intensity or brightness of the sun; the higher the number, the brighter the light.

Shadows

Toggles the display of shadows by the sunlight.

Note: The color of the sunlight is hard-coded.

Sun Angle Calculator

Date

Displays the current date. Edit the date to show the sun at a different time of year.

Time

Displays the current time. Edit the time to show the sun at a different time of day.

Daylight saving

Toggles the effect of daylight saving time on the time property.

Azimuth

Reports the horizontal angle of the sun measured clockwise from the north.

Note: This setting cannot be changed as the program calculates it from the other settings in this section.

Altitude
Reports the vertical angle of the sun, measured from the horizon.

Note: This setting cannot be changed as the program calculates it from the other settings in this section.

Source vector
Reports the direction of the sunlight. This setting cannot be changed as the program calculates it from the other settings in this section.

Rendered Shadow Details

Type
Displays realistic shadows with softer shadows based on extended light sources, according to the **Softness** property.

Softness
Determines the sampling size of shadows. Larger numbers are more accurate but take longer to render.

Geographic Location
Reports location of the drawing on earth. To change the location, click the **Edit** button to access the **Geographic Location** dialog box; see GEOGRAPHICLOCATION command.

24.130.3 Context menu options

New
Loads additional light definitions into the drawing.

Delete
Deletes light definitions from the drawing.

Rename
Renames the light definition.

Select All
Selects all light definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected light definition as current.

24.131 SUPERHATCH command (Express Tools)

Hatches entities using images, blocks, external references or wipeouts as hatch pattern.

Shape Lite Pro Mechanical BIM

Icon: 

24.131.1 Method

Opens the **SuperHatch** dialog box that allows you to select an entity as hatch pattern.

Then, pick an internal point where the hatch has to be applied.

24.132 SUPPORTFOLDER command

Opens the operating system's file manager at the Support folder.

Shape Lite Pro Mechanical BIM

24.132.1 Description

Displays a file manager dialog box showing the contents of the **Support** folder.

Note: The **Support** folder holds many files that drawings and commands need, such as linetype patterns and tool palette libraries.

Note: **Support** folder paths are stored in the SRCHPATH system variable, and the default path is: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*.

24.133 SUPPRESS command

Suppresses the display of entities.

Shape Lite Pro Mechanical BIM

Icon: 

24.133.1 Description

Creates named groups of entities whose display can be turned Off.

Note: You can turn their display back On through the **Mechanical Browser** or **Parameters Manager** panels. See more information in the **Suppression feature** article.

24.133.2 Options within the command

New
Create a new named group of entities.

Note: The group's name cannot begin with digits or special characters.

Edit
Change the entities included with a named group.

Dissolve
Turn the display of suppressed entities back on and remove the name of the group.

24.134 SVGOPTIONS command

Opens the **Settings** dialog box with the **SVG export** category expanded.

Shape Lite Pro Mechanical BIM

24.134.1 Description

Opens the **Settings** dialog box with the **SVG export** category expanded to view and modify relevant system variables.

24.135 SWEEP command

Creates 3D solids or surfaces by sweeping profiles along a path.

Shape | Lite | Pro | Mechanical | BIM

Icon: 

24.135.1 Description

Creates 3D solids or surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path.

Entities accepted as profile entities: 2D splines, 3D splines, 2D polylines, 2D solids, 3D solid faces, 3D solid edges, arcs, circles, ellipses, elliptical arcs, lines, regions, surface and mesh edges, traces.

Entities accepted as a path entity: 2D splines, 3D splines, 2D polylines, 3D polylines, edges of 3D solids, surfaces or meshes, helices, arcs, circles, ellipses, elliptical arcs, lines.

Note: Depending on the value of the DELOBJ system variable, the defining entities are either retained or deleted.

24.135.2 Method

There are two methods to sweep entities:

- Create 3D solids
- Create surfaces

24.135.3 Options within the command

Select profile entities
Allows you to select the profile entities.

Note: Press ENTER after selecting the profile.

Select sweep path
Allows you to select an entity to define the path.

Alignment
Determines the angle at which the profile 2D entities are swept along the paths.

Yes
Aligns the sweep entity perpendicularly to the sweep path entity.

No
The sweep entity maintains its original orientation and is therefore not rotated along the path.

Base point
Defines the point on the sweep entity that follows the path. This option allows offset sweep paths.

Twist
Specifies a twist angle for the swept entity. The twist angle is the amount of rotation along the entire length of the sweep path.

allow Banking for a non-planar path
Determines whether the swept entities will rotate naturally along a 3D sweep path, such as those made from a 3D polyline, spline, or helix.

Profile orientation
Sets the profile orientation for sweep.

Follow
Creates a sweep that follows the path.

Constant
Creates a sweep that keeps profile normal constant.

Create
Creates a sweep entity. This is the default option.

Subtract
Subtracts the sweep entity from the solids or surfaces that intersect with it.

Unite
Unites the sweep entity with the solids or surfaces that intersect with it.

Note: The **Subtract** and **Unite** options are available only for solid sweep entities.

Note:

- If the Hotkey Assistant (HKA) is ON, the Hotkey Assistant widget displays, indicating whether the command is in the **Create**, **Subtract**, or **Unite** mode. Repeatedly press the Ctrl key during the dynamic display of the sweep to cycle the various options.



- The Boolean operations work both in interactive and non-interactive mode, but in interactive mode, they only perform after you finish the interactive sweep.

MOde
Sets the 3D entity type this command will create.

SOlid
Creates 3D solids.

Note: Solid mode is the default one, unless an open contour entity is selected first.

Note:

- Only for **Solid** mode, when the CREATESKETCHFEATURE system variable is ON, sketch features for sweep are created in a dedicated BC_SKETCHES layer, which is not visible by default. The sketch features for sweep (sketches and paths) are visible and editable as block references in the **Mechanical Browser** panel.
- If the edited sketch becomes an invalid geometry, the settings revert to the original state.
- The CREATESKETCHFEATURE system variable can also be controlled by pressing the **CreateSketchFeature** toggle button in the ribbon.

SUrface
Creates surfaces.

Note: Open entities always construct 3D surfaces.

Note: It is not allowed to use a combination of different profiles to create a solid and a surface in one sweep entity. If you select a closed contour profile first, any opened contour profiles will be removed from the selection.

Interactive
Allows to dynamically sweep profiles along a sequence of interconnected linear entities or along a portion of a path.

Transform profile
Use the Manipulator to move, rotate, mirror or scale the profile.

Accept
Accepts the current position of the profile.

Move cursor along selected edge
The profile is swept dynamically along the path or path segment.

Select adjacent edge
Click when the next segment highlights to continue.

Undo
Removes the last segment.

Circle
Creates a sweep entity by selecting a path using a circle profile without a sketch.

Note: The circle plane is normal to the path.

Radius of circle
Defines the radius of the circle profile entity.

Back
Goes back to the previous Command line prompt.

24.136 SYSVDLG command (Express Tools)

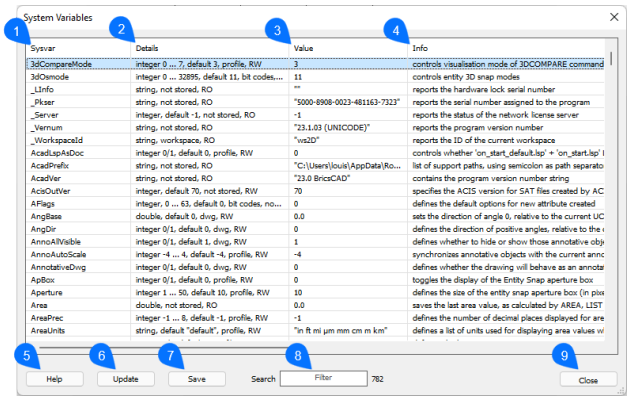
Views, edits and saves system variable settings.

Shape Lite Pro Mechanical BIM

Icon: 

24.136.1 Method

Opens the **System Variables** dialog box that allows you to view, restore, edit and save system variable settings.



1 System variables

2 Details

3 Value

4 Info

5 Help

6 Update

7 Save

8 Search

9 Close

24.136.2 System variables

Lists the system variables.

24.136.3 Details

Lists the details of the system variables. This includes: variable type, minimum and maximum value, default value...

24.136.4 Value

Shows the current value, double click on this value redirects you to the correct dialog box to change it. Lists the details of the system variables.

24.136.5 Info

Lists additional info about the system variable.

24.136.6 Help

Redirects you to the BricsCAD helpdesk.

24.136.7 Update

Refreshes the dialog box.

24.136.8 Save

Saves the system variable settings to a .txt file.

24.136.9 Search

Allows you to search/filter the system variables.

24.137 SYSWINDOWS command

Arranges windows.

Shape Lite Pro Mechanical BIM

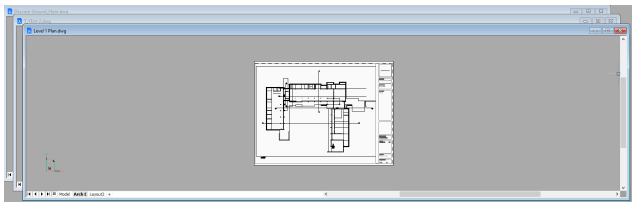
24.137.1 Description

Arranges the program's windows in tiled or cascaded formations (short for "system windows").

24.137.2 Options within the command

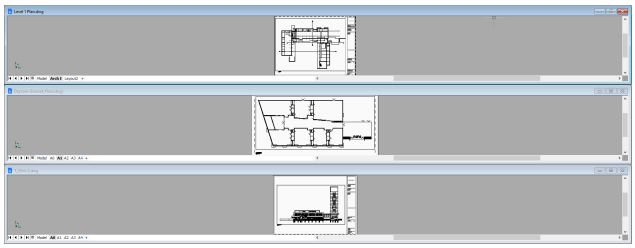
Cascade

Overlaps windows in a cascading arrangement like the WCASCADE command.



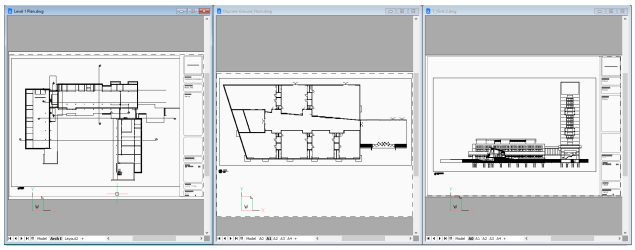
tile Horizontally

Arranges windows horizontally like the WHTILE command.



tile Vertically

Arranges windows vertically like the WVTILE command.



Arrange icons

Arranges icons of minimized windows at the bottom of the main window like the WIARRANGE command.


Lev...
T_Y...
Da...

25. T

25.1 TABLE command

Opens the **Insert table** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

25.1.1 Description

Opens the **Insert Table** dialog box to create a table entity in the current drawing, either from scratch or from data in an external file or a data link.

25.1.2 From scratch

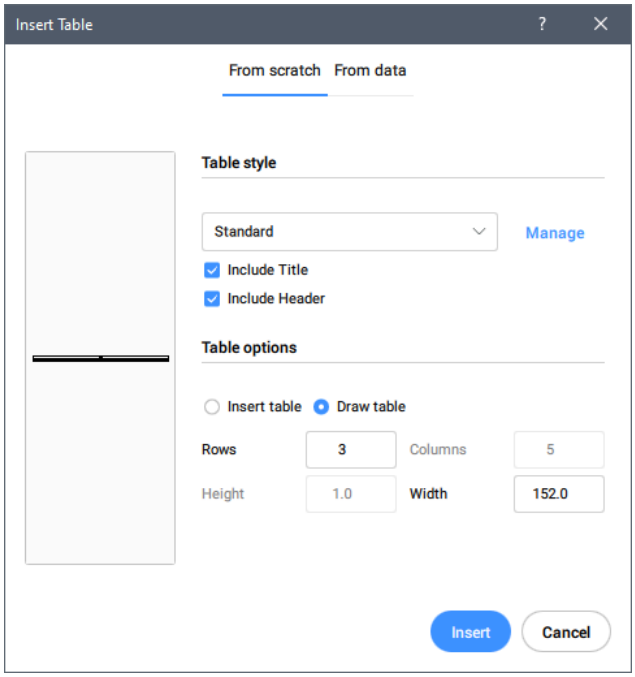


Table Style

Chooses an existing table style from the drop-down list.

Note: The **Manage** button opens **Drawing Explorer** > **Table Styles** .

Table Options

Choose to insert a table by defining its properties or to draw a table.

25.1.3 From data

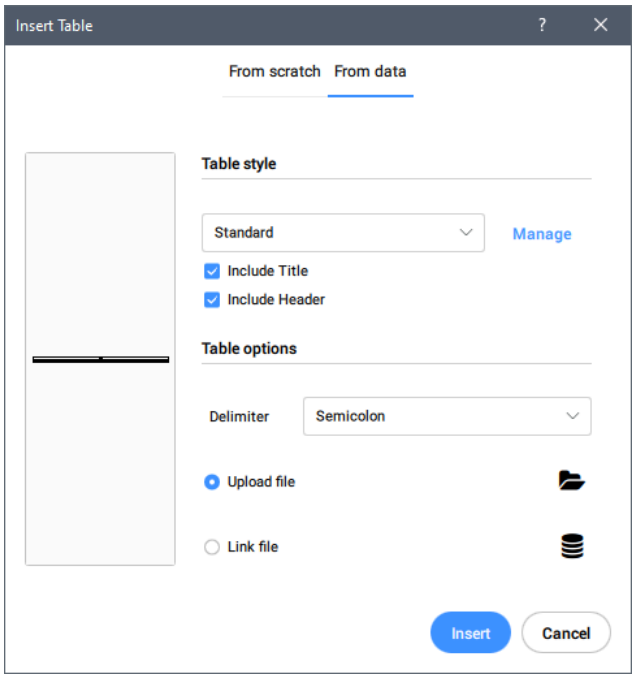


Table options

Delimiter
Specifies the delimiter character used to separate fields in CSV data files. Select the delimiter you want to use from the drop-down list.

Note: Only available for the **File** option.

Upload file
Displays the **Select source file** dialog box to choose a file for the new table.

Link file
Displays the **Datalink Manager** dialog box. See the related article **Datalink Manager dialog box**.

25.2 -TABLE command

Creates a table entity.

Shape Lite Pro Mechanical BIM

25.2.1 Description

Creates a table using specified number of columns and rows or automatically by specifying a point and dragging.

25.2.2 Method

There are 2 methods:

- By specifying the number of columns and rows.
- Automatically by dragging.

25.2.3 Option within the command

Style
Specifies the table style.

Width
Specifies the column width.

Height
Specifies the minimum row height.

Auto
Specify for the columns, rows or both to be created automatically by dragging.

Note: The Text Formatting window is displayed after creating the table.

25.3 TABLEDIT command

Edits text in a table cell.

Shape Lite Pro Mechanical BIM

25.3.1 Method

Edits text in a table cell through the **Text Formatting** window.

Note: To edit the format of cells, use the TABLEMOD command.

Note: The **Text Formatting** window operates the same way as the one from the MTEXT command.

Note: Error cells are evaluated as blank or zero values when used in another cell's formula.

25.4 TABLEEXPORT command

Exports the contents of a table.

Shape Lite Pro Mechanical BIM

25.4.1 Method

Exports the contents of a table to a CSV or XML file, through the **Export Data** dialog box.

Note: The CSV file can be imported into spreadsheet and database programs to format and further process the data.

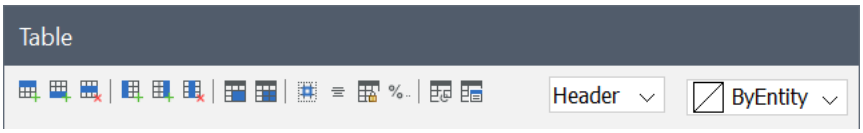
25.5 TABLEMOD command

Formats the properties of table cells and inserts blocks.

Shape Lite Pro Mechanical BIM

25.5.1 Method

Select a table cell to edit and the Table toolbar will be displayed.



Note:

- Hold the left mouse button and define a rectangle to select multiple cells.
- To edit format text in cells, use the TABLEDIT command.

25.5.2 Options within the Table toolbar

Insert Row Above
Inserts an empty row above the selected cell.

Insert Row Below
Inserts an empty row below the selected cell.

Delete Row
Deletes the row(s) of the selected cell(s) without warning.

Note: Use the U command to reverse the change.

Insert Column Left
Inserts an empty column at the left of the selected cell.

Insert Column Right
Inserts an empty column at the right of the selected cell.

Delete Column
Deletes the column(s) of the selected cell(s) without warning.

Note: Use the U command to reverse the change.

Merge Cells
Merges the selected cells. Click the icon and choose an option.

- **Merge All**: merges all selected cells into a single cell.
- **Merge by Row**: merges the selected cells into rows, each with one cell.
- **Merge by Column**: merges the selected cells into columns, each with one cell.

Note: When cells are merged, all their content is erased, except the content of the first cell.

Unmerge Cells
Splits previously merged cells by rows and columns. BricsCAD uses the row numbers (1, 2, 3, etc) and column letters numbers (A, B, C, etc) to know how to split the merged cells into the original set.

Note: When cells are unmerged, the content erased by the **Merge Cells** option is not restored.

Set Border style
Sets the border style of the selection. Displays the **Cell Border Properties** dialog box.

Set Cell alignment
Sets the text alignment of text in the selected cells. Click the icon and choose an option from the drop-down menu.

Note: The content of each selected cell is aligned inside the cell. For example, if you select four cells and then choose Bottom Right, the content in each of the four cells is justified bottom-right.

Toggle Lock/Unlock Cell
Locks and unlocks the content and/or the format of selected cells. Click the icon and choose an option:

- **Unlocked**: the selected cells are unlocked, which means that they can be edited.
- **Locked Content**: the content of the selected cells is locked, but the format of the cells can still be modified.
- **Locked Format**: the format of the selected cells is locked, but the content of the cells can still be edited.
- **Locked Format and Content**: the content and the format of the selected cells are locked.

Note: It is common to lock header cells.

Set Cell Format
Sets the cell format of the selection from the **Table Cell Format** dialog box.

Insert block reference
Inserts a block or a drawing in the selected cell. Click the icon to display the **Insert Block in Cell** dialog box.
The **Insert Block in Cell** dialog box manages the insertion of a block in a table cell (see **Insert block in cell dialog box** related article).

Note:

- Placing a large drawing in a small cell can lead to program instability.
- You can have both text and blocks in a single cell.
- Use the **Manage Cell Content** dialog box to arrange the text and blocks.

Manage cell content
Manages blocks and drawings in cells; click the icon to display the **Manage Cell Content** dialog box.

Cell Style
Sets the sub-style of the selected cells. Click the drop list and choose a sub-style name:

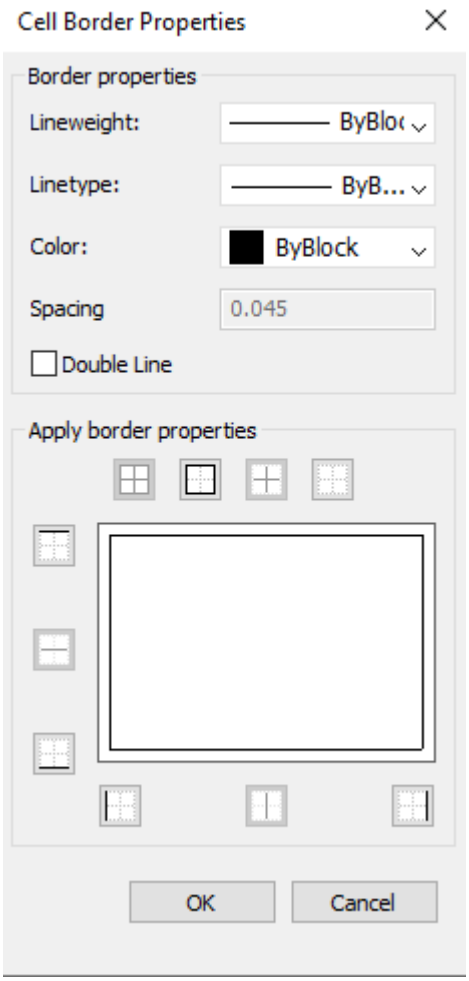
- **Inherit**: the **Cell style** property gets the **By Row/Column** value. **Row style** and **Column style** are available on the **Properties** panel. If they have different styles, the **Column style** takes precedence over the **Row style**.
- **Title**: the title style is usually applied to the topmost row of the table
- **Header**: the Header style is usually applied to the first row of a column
- **Data**: the Data style is usually applied to all other rows in the table

Note: The properties of the Header, Border, and Cell sub-styles are defined by the table style. See the TABLESTYLE command.

Cell Background Color
Sets the background color of the selected cell(s). Click the drop-down list and choose a color.

25.5.3 Cell Border Properties dialog box

The **Cell Border Properties** dialog box controls the border properties of a table cell.



Border properties
Determines the look of the borders that surround each cell. More information about the **Lineweight**, **Linetype**, and **Color** properties can be found in **Properties** panel.

Note: The **Spacing** option is not implemented yet.

Apply border properties
Applies properties to only certain borders.

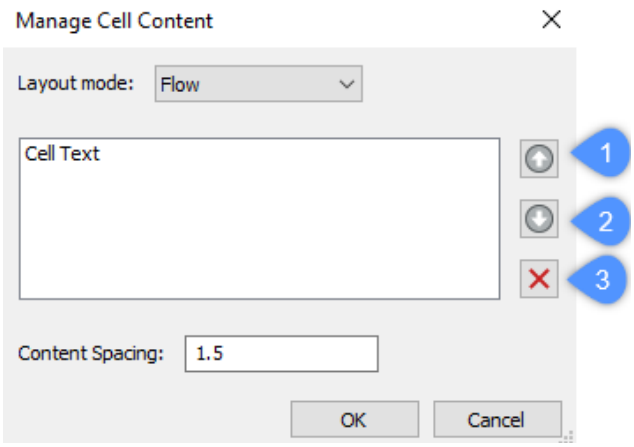
Note:

- Select more than one cell for the Inside borders option to work.
- Although the preview window only shows the double line effect, the other properties are applied to the specified cell border line.

OK
Exits the dialog box, applies the properties and returns to the Table toolbar.

25.5.4 Manage Cell Content dialog box

The **Manage cell Content** dialog box manages the content of a table cell.



Layout mode
When a cell contains more than one element, this option determines how they relate to each other. Click the drop list and then choose an option:

- **Flow**: arranges the elements to fit the cell.
- **Stacked horizontal**: forces the elements to be arranged horizontally.
- **Stacked vertical**: forces the elements to be arranged vertically.

Move Up (1)
Moves the selected element up in the list so that it appears above other elements in the cell.

Move Down (2)
Moves the selected element down in the list so that it's below other elements in the cell.

Delete (3)
Deletes the selected element from the cell.

Content Spacing
Specifies the distance between elements in the cell. Enter a number in drawing units.

Note: This option is inoperable.

OK
Exits the dialog box, changes the arrangement of the cell's contents, and then returns to the Table toolbar.

25.5.5 Context Menu Options

When the TABLEMOD command is running, right-click a cell to open its context menu.

Clear Cell Overrides
Resets the properties of cells to the Cell sub-style settings defined in the table style.

Clear Table Overrides
Resets the table properties to those defined in the table style.

Insert Block Reference
Inserts blocks and drawings.

Manage cell Content
Manages the text and blocks in cells.

Copy
Copies the content of the cell(s) to the Clipboard for pasting into other cell(s).

Cut
Cuts the content of the cell(s) to the Clipboard for pasting into other cell(s).

Paste
Pastes cell content from the Clipboard into cell(s), with these caveats:

- All content in the cell is replaced by the pasted item.
- When more than one cell is selected for pasting, content is pasted only in the first cell.
- Non-cell data is not pasted, such as entities from the drawing or text from another application.

Insert DataLink
Adds data from an external file to the end of the table. Acceptable file formats are:

- CVS: ASCII files in which fields are separated by a delimiter character, such as comma or tab.
- XLS: older Excel spreadsheet files.
- XLSX: new Excel spreadsheet files.

Note: For more details, see the DATALINK command.

Remove DataLink
Removes the link to the table, but the data remains in the table.

Note: Only available when the table has a datalink.

Update DataLink
Updates the data in the table from the external file.

Note: Available only when the table has a datalink.

Size rows equally
Sizes the selected rows to match the size of the tallest cell in the selection set.

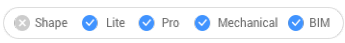
Size columns equally
Sizes the selected columns to match the size of the widest cell in the selection set.

Size all rows equally
Sizes all rows in the table to match the tallest row.

Size all columns equally
Sizes all column in the table to match the widest column.

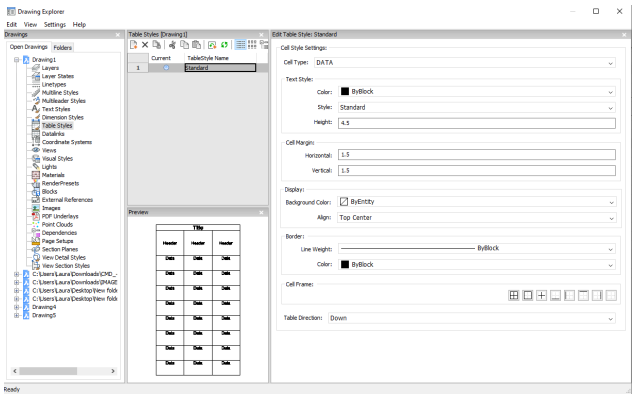
25.6 TABLESTYLE command

Opens the **Drawing Explorer** dialog box with **Table Styles** selected.



25.6.1 Description

Opens the **Drawing Explorer** dialog box with **Table Styles** section selected to view and modify table styles in the current drawing.



25.6.2 Options within the Drawing Explorer

Cell style settings

Defines the cell style settings for each cell style: Title, Header, and Data. Click the drop-down list and choose a cell style.

Text Style

Sets the Color, Style and Height for the selected cell style.

Cell Margin

Defines the spacing between the cell border and the content of the cell.

Horizontal

Specifies the distance between the cell content and the vertical cell borders.

Vertical

Specifies the distance between the cell content and the horizontal cell borders.

Display

Defines the background color and the text alignment.

Background color

Specifies background color This can be done from the drop-down list or through the **Color** dialog box.

Align

Specifies an alignment through a drop-down list

Border

Sets the lineweight and color of the cell borders.

Lineweight

Sets lineweight through a drop-down list.

Color

Pick a border color from the drop-down list, or else choose Select Color to display the **Color** dialog box.

Cell Frame

Applies the border properties to specific lines that make up the cell frames.

Table Direction

Determines the direction in which the table grows when you add and remove rows.

Down
The title and header cells at the top, and the table grows downwards.

Up
The title and header cells at the bottom, and the table grows upwards.

25.6.3 Context Menu Options

New
Creates a new table style.

Delete
Deletes table style definitions from the drawing. The following table style definitions cannot be deleted:

- Table styles in use
- 'Standard' table style

Rename
Renames the select table style.

Select All
Selects all table style definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected table style as current.

25.7 TABLET command

Configures digitizing tablets.

Shape Lite Pro Mechanical BIM

Alias: TA

25.7.1 Description

Works only with Windows, and requires that Wintab32.dll be installed in folder *C:\Windows\System32*.

25.7.2 Method

Toggles tablet mode. Configures and calibrates digitizing tablets.

25.7.3 Options within the command

ON
Turns on digitizing mode so that the entire surface of the tablet can be used for picking points and choosing entities, like a mouse.

OFF
Turns off digitizing mode to make available the menu areas for choosing commands.

CALibrate
Calibrates the corners of the tablet.

ConFiGure
Configures the tablet's menu areas.

25.8 TABSURF command

Creates a 3D mesh surface by extruding an 2D entity along a straight path entity.

Shape Lite Pro Mechanical BIM

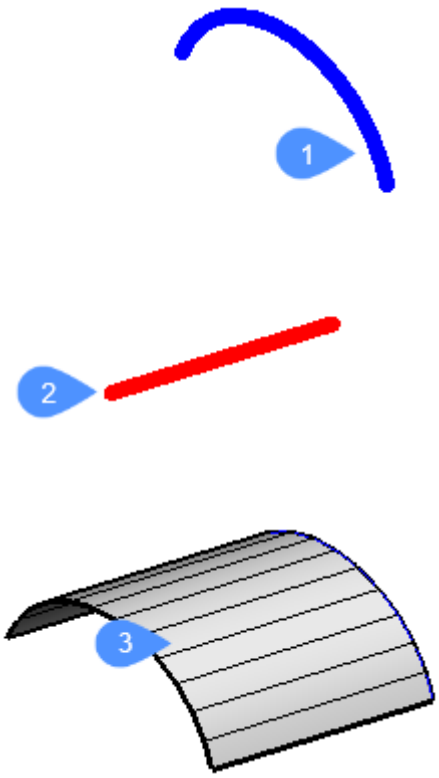
Icon:

25.8.1 Method

Specify the cross-section shape (1) of the extrusion. The entity to be extruded must be open and must be a single entity.

Specify the extrusion path (2), which can be a line or open polyline. The polyline need not be a line segment. If it is a polyarc or multisegment polyline, the extrusion (3), however, is always "straight," going from the start to the endpoint of the polyline.

The path is a vector, which means its length specifies the length of the surface. Its direction specifies the direction of extrusion. The extrusion starts at the entity to be extruded, but goes in the direction indicated by the path. The end selected on the polyline or line defines the direction of the extrusion.



25.9 TANGENT command

Toggles the **Tangent** entity snap.

Shape Lite Pro Mechanical BIM

Icon:



25.9.1 Description

Toggles the **Tangent** entity snap to enable or disable snapping to tangent. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

25.10 TCASE command (Express Tools)

Changes the case of the selected text entities.

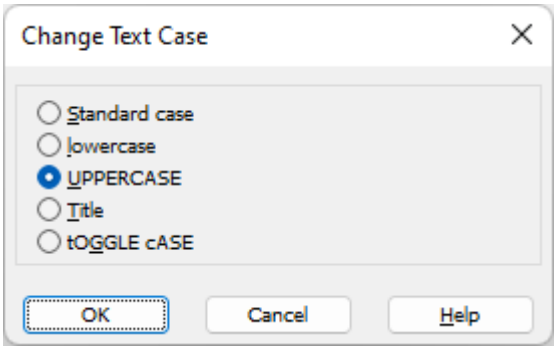
Shape Lite Pro Mechanical BIM

Icon: 

25.10.1 Method

Opens the **Change Text Case** dialog box that allows you to change the case of selected text entities.

The **Change Text Case** dialog box allows you to change the case of selected text entities. You can choose between **Standard case**, **lowercase**, **UPPERCASE**, **Title** and **tOGGLE cASE**.



Standard case
Capitalizes the first letter of the first word.

lowercase
Changes all letters of the selected text to lowercase.

UPPERCASE
Capitalizes all letters of the selected text.

Title
Capitalizes the first letter of each word.

tOGGLE cASE
Changes the first letter of each word to lowercase and capitalizes the rest of the letters.

25.11 -TCASE command (Express Tools)

Changes the case of the selected text entities via the Command line.

Shape Lite Pro Mechanical BIM

25.11.1 Method

Select the text entities you want to change and specify the case mode.

25.11.2 Options withing the command

Sentence
Capitalizes the first letter of the first word.

Lower
Changes all letters of the selected text to lowercase.

Upper case
Capitalizes all letters of the selected text.

Title
Capitalizes the first letter of each word.

toGgle
Changes the first letter of each word to lowercase and capitalizes the rest of the letters.

25.12 TCIRCLE command (Express Tools)

Surrounds text, mtext or attribute definition entities with circles, rectangles or slots.

Shape Lite Pro Mechanical BIM

Icon: Ⓣ

25.12.1 Method

Select the entities and enter an offset factor.

25.12.2 Options within the command

Circles
Draws a circle around the selected text.

Slots
Draws a slot around the selected text.

Rectangles
Draws a rectangle around the selected text.

Constant
Surrounds selected entities with entities that are all the same size, the size of the largest enclosing entity.

Enter constant slot
Allows you to choose between **Width**, **Height** and **Both**.

Note: This option is available only for **Slots** and **Rectangles**.

Variable
Adjusts to the size of each selected entity, based on the offset factor.

25.13 TCONNECT command

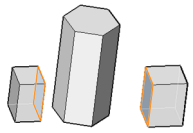
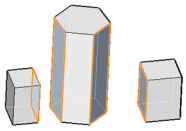
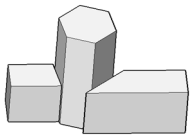
Connects solid faces and edges of planar surfaces to other solids, planar surfaces or regions.

Shape Lite Pro Mechanical BIM

Icons:

25.13.1 Description

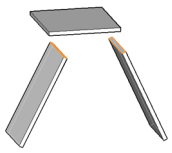
Connects the selected solid faces or edges of the planar surfaces to other solid faces, planar surfaces or regions.

Step1	Step2	Step3
		

Note: If the KEEPCONNECTIONS system variable is ON, the connected solid is subtracted from interfering solids.

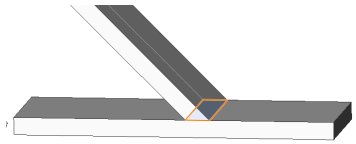
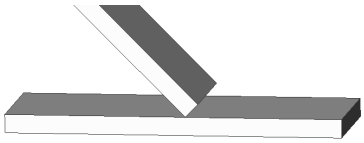
25.13.2 Options within the command

Connect to nearest
Connects the selected face(s) or edges of planar surfaces to the nearest regions, planar surfaces or face(s) of other solid(s).





Disconnect
This option applies to solid faces only. The face is disconnected and made perpendicular to the adjacent faces, which results in a single edge or vertex connection.
Select a 3D solid face, planar surface or region to disconnect from the nearest face in the model (or the currently connected face).

Before	Disconnect result
	

Connect
Switch to the connect face mode.

25.14 TCOUNT command (Express Tools)

Adds a sequential numbering to text and mtext entities.

Shape Lite Pro Mechanical BIM

Icon:

25.14.1 Method

1 Select text entities.

2 Specify the sorting method.

3 Enter starting number and increment, separated by comma.

4 ***Note:*** Negative increments are supported.

Note: Sequential numbering is added to each line of mtext entities and to individual text entities.

25.14.2 Options within the command

X
Adds numbering by increasing the x-coordinate values of the selected text entities.

Y
Adds numbering by decreasing the y-coordinate values of the selected text entities.

Select-order
Adds numbering according to the order in which the text entities have been selected.

Overwrite
Replaces the selected texts with a sequential numbering.

Prefix
Adds numbering in front of the text string.

Suffix
Adds numbering after the text string.

Find&replace
Replaces a specified text string with a sequential number.

25.15 TEMPLATEFOLDER command

Accesses the folder holding template files.

Shape Lite Pro Mechanical BIM

25.15.1 Description

Opens the Templates folder in which the program stores DWT template files; the folder is displayed by the operating system's file manager. To open the template file as a new drawing, double-click a file name.

By default, the template files are found here: *C:\Users\%username%\AppData\Local\Bricsys\BricsCAD\V25x64\en_US\Templates*.

25.16 TEXT command

Creates a single-line text entity.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: TX

Note: When the TEXTEVAL system variable is set to 1, this command evaluates LISP expressions.

25.16.1 Method

There are two methods:

- Places single lines of text in the drawing by specifying height of text and rotation angle.
- Evaluates LISP expressions.

Note: If the text style is annotative, the height value defines the paper space height of the text. The model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

25.16.2 Options within the command

use defined Style
Changes the text style to use for the text.

?
Lists all styles defined in the current drawing.

Align on line
Aligns text between two points.

Note: This option suppresses the height and angle prompts, because the height and angle are determined by the two pick points.

Fit between points
Fits text between two points and prompts for the height.

Note: This option suppresses the angle prompt, because the angle is determined by the two pick points. Entering certain heights can cause the text to be excessively elongated or squeezed.

Center horizontally
Centers text in the middle of the baseline.

Middle (hor/vert)
Centers text on the middle of the text.

Right justify
Right-justifies each line of text.

Justification options...
Displays all justification options. Choose between: Top Left, Top Center, Top Right, Middle Left, Middle Center, Middle Right, Bottom Left, Bottom Center, Bottom Right.

Note: Text can be moved or stretched through grips.

25.17 -TEXT command

Places single-line text in drawings.

Shape Lite Pro Mechanical BIM

Alias: -T

25.17.1 Description

Places single lines of text in the drawing, unformatted; works at the Command line.

See the TEXT command.

25.18 TEXTEDIT command (Express Tools)

Edits the selected text entity.

Shape Lite Pro Mechanical BIM

25.18.1 Description

Opens the In-Place text editor to edit the selected text entity.

25.19 TEXTFIT command (Express Tools)

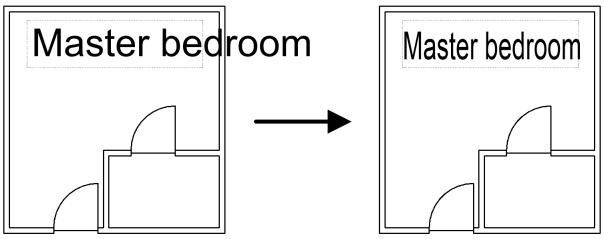
Fits text between two selected points.

Shape Lite Pro Mechanical BIM

Icon: 

25.19.1 Method

Select a text to stretch or shrink and specify the end point.



25.19.2 Options within the command

Start point
Allows you to specify a new starting point for the selected text entity.

25.20 TEXTMASK command (Express Tools)

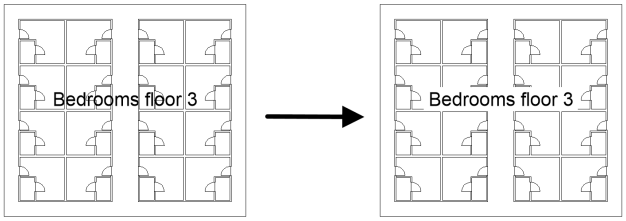
Creates a mask behind the selected text or mtext entities.

Shape Lite Pro Mechanical BIM

Icon: T

25.20.1 Method

Creates a mask entity behind the selected text or mtext entities, by specifying the mask type and the offset value.



Note: Use the TEXTUNMASK command to remove the mask from the text.

25.20.2 Options within the command

Masktype
Allows you to set the mask type.

Wipeout
Creates a wipeout entity, displayed in the background color of the drawing.

3d face
Creates a 3D face entity.

Solid
Creates a 2D solid entity, using a specified color.
Opens the **Color** dialog box which allows you to select the color.

Offset
Allows you to define an offset distance.

25.21 TEXTSCR command

Switches to the text screen.

Shape Lite Pro Mechanical BIM

25.21.1 Description

Displays the Prompt History window (short for "text screen"). This window displays the last 256 lines of command and prompt text. To display greater or fewer lines of history, change the value of the SCRLHIST variable. To return to the graphics screen, enter the GRAPHSCR command or press F2.

Note: A context menu displays when you right click on the command history. See the full description in the Command line panel article.

25.22 TEXTTOFRONT command

Displays texts, dimensions, and/or all leaders on front of all other drawing entities.

Shape Lite Pro Mechanical BIM

Icon: 

25.22.1 Description

Brings the display of all texts, all dimensions, and/or all leaders to the front of the display order, so that no other drawing entities can overlap texts, dimensions, and leaders.

25.22.2 Options within the command

Text
Brings all texts to the front of the display order.

Dimensions
Brings all dimensions to the front.

Leaders
Brings all leaders to the front.

All
Brings texts, dimensions, and leaders to the front.

25.23 TEXTUNMASK command (Express Tools)

Removes the mask from behind selected text or mtext entities.

Shape Lite Pro Mechanical BIM

Icon: 

25.24 TFRAMES command (Express Tools)

Toggles visibility of image and wipeout frame borders.

Shape Lite Pro Mechanical BIM

Icon:

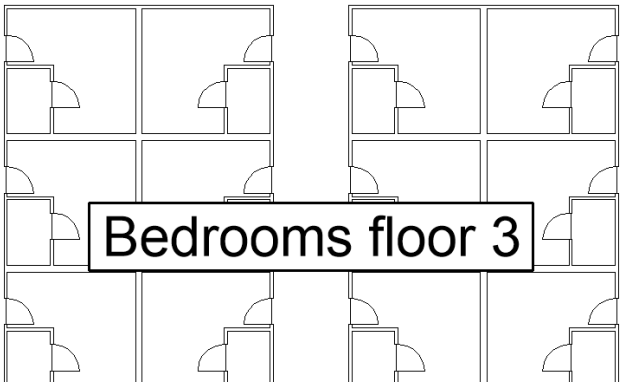
25.24.1 Method

Automatically changes the frame display of wipeouts and images in the drawing.

- **TFRAMES is OFF**:



- **TFRAMES is ON**:



25.25 TIFFOUT command

Saves the selected entities to a TIF file.

Shape Lite Pro Mechanical BIM

25.25.1 Method

Opens the **Save TIFF** dialog box that allows you to choose a name and a folder for the TIF file.

Select the entities you want to include in the TIF file.

Note: The TIF file reflects what is shown on the screen.

25.26 TIME command

Reports the use of time in the drawing.

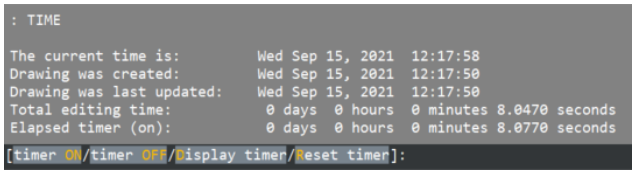
Shape Lite Pro Mechanical BIM

Icon:

Alias: TI

25.26.1 Description

Reports the date and time the drawing was created, last updated, and spent editing; works at the Command line.



25.26.2 Options within the command

timer ON
Turns on the independent (“elapsed”) timer. The program reports, “The drawing timer is ON.”.

timer OFF
Turns off the independent timer. The program reports, “The drawing timer is OFF.”.

Display timer
Reports the date and time the drawing was created, last updated, and the amount of time spent editing (that the drawing was open).

Reset timer
Resets the independent timer.

25.27 -TIN command

Creates a new TIN Surface from selected source data via Command line.

Shape Lite Pro Mechanical BIM

25.27.1 Description

A TIN Surface is a three-dimensional geometric representation of a terrain or a civil feature, which also includes its representation with triangles, contours, slopes, spot elevations, and elevation ranges.

The algorithm connects the nearest lying points into triangles that form a Triangular Irregular Network (TIN), using the Delaunay triangulation method.

You can affect the way points are connected by adding boundaries and breaklines to the TIN Surface. Triangles always follow specified boundaries and breaklines and never intersect them. You can also edit the TIN Surface with other options available in the TINEDIT command.

A TIN Surface is dynamically linked to most of its input data, such as Point files, Point Groups, Point Clouds, Civil points and Contours. Point files and point clouds are usually no longer modified by users after a TIN Surface has been created from them. Therefore, in the next step after the creation of the TIN surface, a snapshot of the TIN surface is automatically created. This breaks the link to the input data. You can remove the **Create snapshot** definition in the **Civil Explorer** to re-establish the dynamic link.

Dynamic link is not supported for CAD elements and 3D faces. Changing these types of input data does not affect the triangulation of the TIN surface.

25.27.2 Method

The following type of source data can be used to create a TIN Surface with the TIN command:

- CAD elements (points and linear entities) and Civil points
- Point files
- Civil points specified in the drawing
- 3D Face elements
- Point Clouds
- Point Groups
- Contours

Launch the TIN command from the Ribbon, Civil Toolbar, Command line, or from the **Civil Explorer** panel. Select the source data type and specify the input elements. New data is added to the TIN Surface as a new Definition.

25.27.3 Options within the command

Import from file
Creates a TIN surface from an imported points file in text file format (TXT), comma-delimited file format (CSV) and other text formats where the XYZ coordinates in the input file are separated by any delimiter. You can select multiple point files to create a TIN surface. The file names and their path have to be introduced in the Command line.

Enter file format name
Sets the file format name.

? to list point file formats
Lists the names of all point file formats.

place Points
Creates a TIN surface from picked points.

create from Faces
Creates a TIN Surface by selecting 3D faces. You can choose to add edges of 3D faces as breaklines (Y) or to create TIN Surface only with points of 3D Faces (N).

create from point Cloud
Creates a TIN Surface from a Point Cloud.

create from point Groups
Creates a TIN Surface from a Point Group.

create from cOntours
Creates a TIN Surface from selected line elements.

Use weeding factors
Yes: existing contour vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.
No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

Use supplementing factors
Yes: additional vertices are added to contours, if the distance between vertices on the contour is greater than the specified supplementing distance.
No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

cLip polygon
Creates a TIN Surface only in the area defined by the CLIP polygon (a closed polyline).

Note: An option exists to remove the crossed breaklines, if any.

Simplify the TIN surface?
Yes: the TIN Surface created from an imported points file is simplified, and from a Point Cloud by reducing the points density. The surface simplification process is defined by setting values for the radius or simplification step, and for elevation difference.
No: a TIN Surface is created from all points in the input point file or in Point Cloud.

Note: After the first iteration of the simplification, a report on the command line displays, stating the number of points before simplification, the number of removed points and the number of points after simplification. In the next step, you can choose between the options **Simplify again** or **Create TIN Surface**.

Create TIN surface
Creates the TIN Surface.

Densify TIN surface
Yes: after the TIN Surface is simplified, the maximum elevation difference between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface is specified. All input points with a greater elevation difference than specified are added back to the simplified TIN Surface.
No: the TIN surface is created according to the specified parameters for simplification.

Enter for entire TIN surface
Applies the densify process for the entire TIN Surface.

Select polygons
Applies the densify process for the selected polygons.

Draw polygon
Applies the densification for a polygon, specified in a drawing.

Note: The parameters for TIN Simplify and TIN Densify processes can be subsequently edited using the appropriate definition for creating a TIN surface in **Civil Explorer**.

25.28 TIN command

Creates a new TIN Surface from selected source data.

Shape Lite Pro Mechanical BIM

Icon:

25.28.1 Description

A TIN Surface is a three-dimensional geometric representation of a terrain or a civil feature, which also includes its representation with triangles, contours, slopes, spot elevations, and elevation ranges.

The algorithm connects the nearest lying points into triangles that form a Triangular Irregular Network (TIN), using the Delaunay triangulation method.

You can affect the way points are connected by adding boundaries and breaklines to the TIN Surface. Triangles always follow specified boundaries and breaklines and never intersect them. You can also edit the TIN Surface with other options available in the TINEDIT command.

A TIN Surface is dynamically linked to most of its input data, such as Point files, Point Groups, Point Clouds, Civil points and Contours. Point files and point clouds are usually no longer modified by users after a TIN Surface has been created from them. Therefore, in the next step after the creation of the TIN surface, a snapshot of the TIN surface is automatically created. This breaks the link to the input data. You can remove the **Create snapshot** definition in the **Civil Explorer** to re-establish the dynamic link.

Dynamic link is not supported for CAD elements and 3D faces. Changing these types of input data does not affect the triangulation of the TIN surface.

25.28.2 Method

The following type of source data can be used to create a TIN Surface with the TIN command:

- CAD elements (points and linear entities) and Civil points
- Point files
- Civil points specified in the drawing
- 3D Face elements
- Point Clouds
- Point Groups
- Contours

Launch the TIN command from the Ribbon, Civil Toolbar, Command line, or from the **Civil Explorer** panel.

Select the source data type and specify the input elements. New data is added to the TIN Surface as a new Definition.

25.28.3 Options within the command

Select entities to create TIN surface

Creates a TIN Surface from selected drawing entities/objects.

When creating TIN Surface from linear objects, the Command line prompts you to choose an option how to apply linear entities: as **Points**, **Edges** or **Breaklines**.

If you select to apply linear entities as breaklines, a new **Add breaklines** definition is added to the TIN Surface. In this case, the Command line prompts you to specify the weeding and supplementing parameters. Some other parameters, such as the Breakline type, the Crossing elevation and the Mid-ordinate distance, are pre-defined. They can be easily modified in the **Add breaklines** definition dialog box from the **Civil Explorer** panel.

Otherwise, the selected linear objects are added to the **Add drawing objects** definition as TIN Surface points or edges.

Import from file

Creates a TIN surface from selected point files. Selecting this option in the Command line opens the **Create Surface From Point File** dialog box, where you can select point files and appropriate point file formats. If the predefined point file formats are not suitable, you can define your own format with a custom file

extension, delimiter, comment tag and columns. When a file is selected, the command offers the option to simplify and densify points used to create the TIN Surface.

Note: For more information about TIN Definitions see **TIN Definitions in the TIN command** and **Civil Explorer panel** articles.

place Points
Creates a TIN surface from specified locations and elevations of the TIN points.

Pick point
Allows you to specify the location of the TIN point.

Enter elevation
Allows you to enter the elevation of the TIN point.

create from Faces
Creates a TIN Surface from the selected 3D Face elements.
If you select the **Apply 3D Faces as edges and visibility** option, the vertices and edges of 3D Face elements are taken for the triangulation. Thus, TIN Surface triangles overlap with the 3D faces. The shape of the outer envelope of the 3D face elements is also preserved. Otherwise, only the vertices of the 3D Face elements are taken for triangulation. This means that the TIN Surface does not always overlap with 3D Face elements. Also, additional triangles are created on the outer envelope of the 3D Face elements so that the outer boundary of the TIN Surface has a convex shape.

create from point Cloud
Creates a TIN Surface from a selected Point Cloud.

Simplify TIN surface?
Yes: simplifies source points used to create TIN Surface based on the specified simplification parameters, described below.

- **Radius for simplification step**: the cell size, that forms virtual grid spread over the entire set of source points. Within each cell, all points with an elevation difference greater than the specified maximum, are eliminated / simplified.
- **Elevation difference**: maximum elevation difference between adjacent points on the TIN Surface. All points with a higher elevation difference are excluded from the TIN Surface.

No: a TIN Surface is created from all points in the input point file or in Point Cloud.

Note: After the first iteration of the simplification, a report on the command line displays, stating the number of points before simplification, the number of removed points and the number of points after simplification.

In the next step, you can choose between the options Simplify again or Create TIN Surface.

Create TIN surface
Creates the TIN Surface.

Simplify again
Simplifies the TIN Surface with new proposed simplification parameters so that more points are removed than in the first iteration.

Densify TIN surface
Yes: after the TIN Surface is simplified, the maximum allowed **elevation difference** between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface is specified. All input points with a greater elevation difference than specified are added back to the simplified TIN Surface.

No: the TIN surface is created according to the specified parameters for simplification parameters.

Select polygons
Densifies the TIN Surface within the selected polygon.

Draw polygon
Densifies the TIN Surface within the drawn polygon.

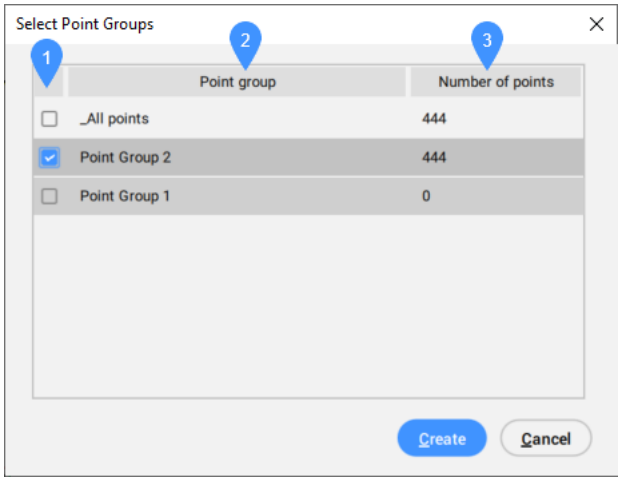
Enter for entire TIN surface
Densifies the entire TIN Surface.

Note: For more information on the simplify and densify process, see the **TIN Simplify** and **TIN Densify** articles.

create from point Groups
Creates a TIN Surface from a selected Point Groups.

Note: By default, Civil points are placed in the "_All points" Point Group. Use the CIVILPOINTGROUP command to create a new Point Group.

The **Select Point Groups** dialog box opens:



1 Select box

2 Point group

3 Number of points

Select box
Allows you to select or deselect a point group to be included.

Point group
Lists all available point groups.

Number of points
Displays the number of points in the point group.

create from cOntours
Creates a TIN Surface from selected line elements.

Use weeding factors

Yes: existing contour vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.

No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

Use supplementing factors

Yes: additional vertices are added to contours, if the distance between vertices on the contour is greater than the specified supplementing distance.

No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

create from Mesh

Creates a TIN Surface from Mesh or Polyface Mesh entities.

cLip polygon

Adds a Clip Boundary to the TIN Surface.

create from point Groups

Creates a TIN Surface from a selected Point Groups.

create from cOntours

Creates a TIN Surface from selected line elements.

Use weeding factors

Yes: existing contour vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.

No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

Use supplementing factors

Yes: additional vertices are added to contours, if the distance between vertices on the contour is greater than the specified supplementing distance.

No: all the vertices of source line elements are used to create the TIN Surface from the Contours.

cLip polygon

Adds a Clip Boundary to the TIN Surface.

Note: For more information about clip polygon see **TIN Definitions in TINEDIT command** article.

25.29 TINASSIGNIMAGE command

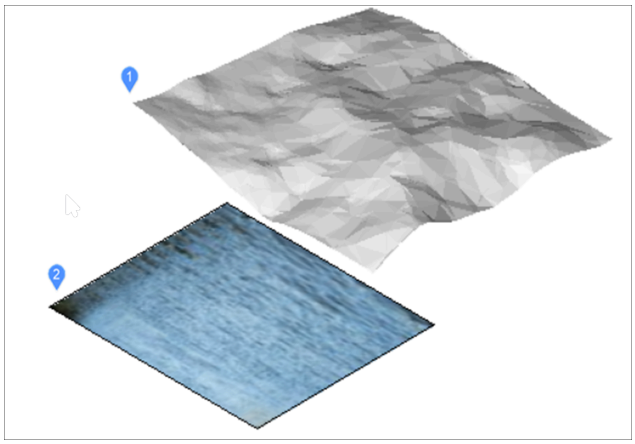
Assigns a raster image as a TIN surface material.

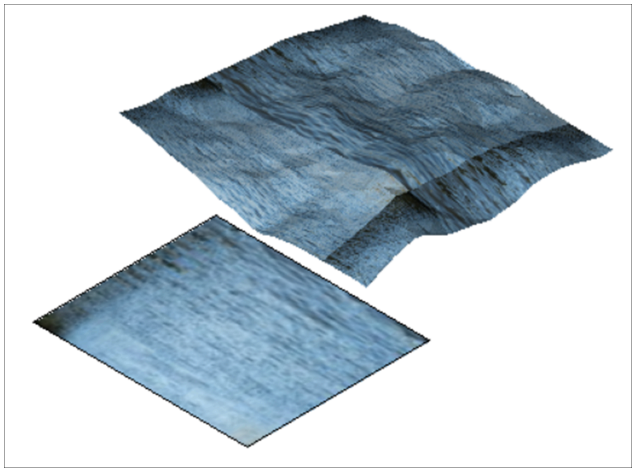
Shape Lite Pro Mechanical BIM

Icon: 

25.29.1 Method

Select the TIN surface (1) and select an attached raster image (2) to assign to the surface.





Note: When you assign a very large raster image to a TIN surface, a warning message is displayed that offers a way to automatically create a reduced raster.

25.30 -TINEDIT command

Edits a TIN surface via Command line.

Shape Lite Pro Mechanical BIM

25.30.1 Description

Allows you to edit existing TIN Surfaces. Editing options affect the way TIN edges connect TIN points to form triangles. You can also delete existing TIN points and edges or add new ones or change the position and elevation of existing TIN points.

Any TIN Surface editing operation can be modified in the **Civil Explorer** panel after the execution. It is also possible to change the chronological order of editing operations.

25.30.2 Options within the command

Add Points

Adds new points to the existing TIN Surface.

Note: These entities are added to the TIN Surface as drawing objects definitions.

Select points or linear entities
Selects points or linear entities to add to the TIN surface.

Pick points
Allows you to specify the location of the TIN point.

Enter elevation
Allows you to enter the elevation of the TIN point.

Add Point Groups
Adds Civil points from a specified Point Groups to the existing TIN Surface.

Note: By default, Civil points are placed in the "_All points" Point Group. Use the CIVILPOINTGROUP command to create a new Point Group.

Enter Point Groups names
Allows you to enter the Point Groups names, separated by comma.

Add Breaklines
Adds selected linear CAD objects to the TIN Surface as breaklines.

Select breakline type
Allows you to choose the breakline type.

Standard
Adds breaklines to the TIN Surface, re-builds the TIN Surface so that triangles follow the breaklines edges and elevations.

Projected
Projects breaklines to the TIN Surface and creates new triangles, without modifying TIN Surface elevations.

seTtings
Allows you to choose the settings for the **Standard** breakline type only.

Use weeding factors?
Toggles the use of the weeding factor.

Enter weeding distance
Sets the weeding distance.

Enter weeding angle
Sets the weeding angle.

Note: Existing breakline vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.

Use supplementing factors?
Toggles the use of the supplementing factors.

Enter supplementing distance
Sets the supplementing distance.

Note: Additional vertices are added to breakline, if the distance between vertices on the breakline is greater than the specified supplementing distance.

Add bOundaries
Adds selected linear CAD objects to the TIN Surface as boundaries.

Select boundary type
Allows you to choose the boundary type.

Show
Displays triangles inside the boundary polygon.

Hide
Hides triangles inside the boundary polygon.

Outer
Defines the outer boundary of the TIN Surface. All the triangles outside the boundary are hidden.

Trim triangle lines

Yes
Trims the triangles (TIN edges) that intersect the boundary to follow the boundary polygon.

No
Deletes all triangles that intersect the boundary polygon. This is the default option.

Clip
Clips an existing TIN Surface in the drawing with a selected polygon as the clipping boundary for the surface.
Does not change the TIN Surface boundary but affects all elements (TIN operations) added to the TIN Surface after the Clip boundary. Only elements located inside the Clip boundary affect triangulation.

Remove Points
Removes points/vertices of triangles from the TIN Surface.

Multiple
Allows you to remove multiple TIN points at once.

Swap Edge
Swaps the selected TIN edges.

Note: Edges cannot be swapped in following cases:

- when the TIN edges lie on the breaklines.
- when the TIN edges are created with the **Add line** definition.
- when the TIN edge belongs to two adjacent triangles that form a concave envelope.

Point Location
Modifies the location of a single or multiple TIN points in one step.

Multiple
Requires a polygonal selection of multiple TIN points and specifying a base point and second point for moving the TIN points.

Point Elevation
Modifies the elevation of a single or multiple TIN points in one step.

Multiple
Requires a polygonal selection of multiple TIN points. Then determine the new absolute elevation or delta elevation (elevation difference).

Add Line
Adds new TIN Edge between existing TIN points.

Delete Line
Deletes TIN Edges inside the specified selection area.

Note: The edges inside the selected area are colored red.

Remove Outer Edges
Removes triangles on the outer boundary of the TIN Surface, that have:

- greater edge length.
- shorter edge length.
- greater angle than specified.

Note: You can specify which of these criteria is considered when removing the outer edges.

This method begins removing triangles from the outer boundary towards the inner part of the TIN Surface. It stops removing triangles, when the last outer triangle meets the specified criteria. That means that this method does not remove inside triangles.

Use maximum triangle length
Toggles the use of the maximum triangle length.

Maximum triangle length
Allows you to specify the maximum triangle length.

Use minimum triangle length
Toggles the use of the minimum triangle length.

Minimum triangle length
Allows you to specify the minimum triangle length.

Use maximum angle
Toggles the use of the maximum angle between adjacent TIN lines.

Maximum angle between adjacent TIN lines
Allows you to specify the maximum angle between adjacent TIN lines.

Minimize Flat Areas
Finds flat triangles and then swaps the adjacent TIN edges of those triangles so that their slope is non-zero.
This method is most useful when the TIN Surface is created from Contours. It starts at the concave side of the contours and swaps the edge of the first triangle so that its slope becomes non-zero. The process continues until the method still ensure that there are no flat triangles by swapping their edges.

Note: Minimize flat areas method swipes only the TIN edges that do not lie on breaklines.

Remove Elevations
Removes TIN points below/above the specified elevation and displays the number of points removed.

Remove points below elevation
Toggles the option to remove TIN points below a specified elevation.

Remove points with elevation less than
Sets the elevation below which TIN points are removed.

Remove points above elevation
Toggles the option to remove TIN points above a specified elevation.

Remove points with elevation greater than
Sets the elevation above which TIN points are removed.

SMoothing
Allows you to smooth a TIN surface by adding additional points.

Enter grid X spacing
Allows you to set the grid X spacing value.

Enter grid Y spacing
Allows you to set the grid Y spacing value.

Enter grid angle
Sets the grid angle value. You can also define it by picking the angle in the drawing area.

Entire TIN surface

Yes
Selects the entire TIN surface.

Pick polygon
Allows you to define the TIN surface area for smoothing by picking a polygonal selection window.

25.31 TINEDIT command

Edits a TIN surface.

Shape Lite Pro Mechanical BIM

Icons: 

25.31.1 Description

Allows you to edit existing TIN Surfaces. Editing options affect the way TIN edges connect TIN points to form triangles. You can also delete existing TIN points and edges or add new ones or change the position and elevation of existing TIN points.

Any TIN Surface editing operation can be modified in the **Civil Explorer** panel after the execution. It is also possible to change the chronological order of editing operations.

25.31.2 Options within the command

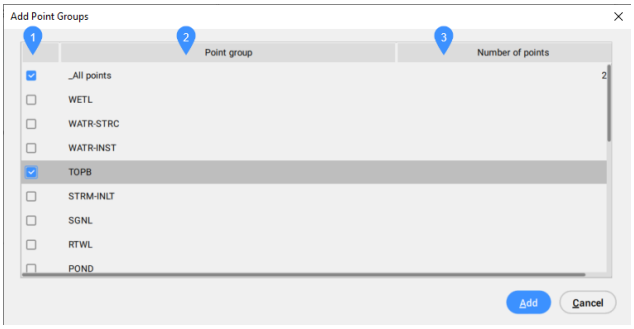
Add Points
Adds new points to the existing TIN Surface.

Note: These entities are added to the TIN Surface as drawing objects definitions.

Add Point Groups
Adds Civil points from a specified Point Groups to the existing TIN Surface.

Note: By default, Civil points are placed in the "_All points" Point Group. Use the CIVILPOINTGROUP command to create a new Point Group.

A dialog box opens:



1 Select box

2 Point group

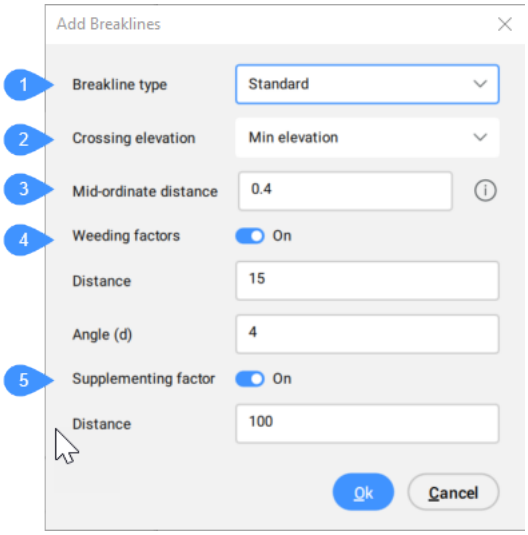
3 Number of points

Select box
Allows you to select or deselect a point group to be included.

Point group
Lists all available point groups.

Number of points
Displays the number of points in the point group.

Add Breaklines
Adds selected linear CAD objects to the TIN Surface as breaklines.
A dialog box opens:



1 Breakline type

2 Crossing elevation

3 Mid-ordinate distance

4 Weeding factor

5 Supplementing factor

Breakline type (1)
Displays the selected breakline type. Select the appropriate breakline type from the drop-down list.

Standard
Adds breaklines to the TIN Surface, re-builds the TIN Surface so that triangles follow the breaklines edges and elevations.

Projected
Projects breaklines to the TIN Surface and creates new triangles, without modifying TIN Surface elevations.

Crossing elevation (2)
Determines which elevation is taken for triangulation at the intersection of two breaklines, added in the same **Add breakline** definition.

Not allowed
The intersection is treated as one breakline.

Min elevation
The elevation of lower breakline is taken.

Mid elevation
The middle elevation between the crossing breaklines is taken.

Max elevation
The elevation of upper breakline is taken.

Mid-ordinate distance (3)
Adds supplementary TIN points along the arc according to the mid-ordinate distance which is used for arc approximation.

Weeding factor (4)
Toggles the use of the weeding factor.

Distance
Sets the weeding distance.

Angle (d)
Sets the weeding angle.

Note: Existing breakline vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.

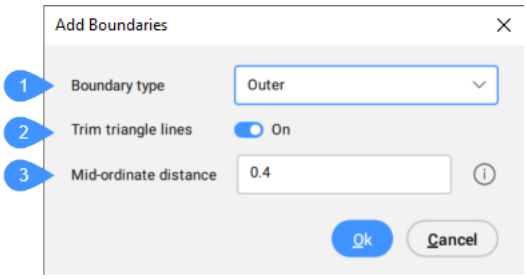
Supplementing factor (5)
Toggles the use of the supplementing factor.

Distance
Sets the supplementing distance.

Note: Additional vertices are added to breakline, if the distance between vertices on the breakline is greater than the specified supplementing distance.

Note: The **Supplementing** and **Weeding factor** are disabled for the **Projected breaklines**.

Add bOundaries
Adds selected linear CAD objects to the TIN Surface as boundaries.
A dialog box opens:



1 Boundary type

2 Trim triangle lines

3 Mid-ordinate distance

Boundary type (1)
You can choose the boundary type from the drop-down menu:

Show
Displays triangles inside the boundary polygon.

Hide
Hides triangles inside the boundary polygon.

Outer
Defines the outer boundary of the TIN Surface. All the triangles outside the boundary are hidden.

Note: Triangle lines are not trimmed by default.

Clip
Clips an existing TIN Surface in the drawing with a selected polygon as the clipping boundary for the surface.
Does not change the TIN Surface boundary but affects all elements (TIN operations) added to the TIN Surface after the Clip boundary. Only elements located inside the Clip boundary affect triangulation.

Trim triangle lines (2)
Yes: trims the triangles (TIN edges) that intersect the boundary to follow the boundary polygon.
No: deletes all triangles that intersect the boundary polygon.

Mid-ordinate distance (3)
Adds supplementary TIN points along the arc according to the mid-ordinate distance which is used for arc approximation.

Remove Points
Removes points/vertices of triangles from the TIN Surface.

Multiple
Allows you to remove multiple TIN points at once.

Swap Edge
Swaps the selected TIN edges.

Note: Edges cannot be swapped in following cases:

- when the TIN edges lie on the breaklines.
- when the TIN edges are created with the **Add line** definition.
- when the TIN edge belongs to two adjacent triangles that form a concave envelope.

Point Location
Modifies the location of a single or multiple TIN points in one step.

Multiple
Requires a polygonal selection of multiple TIN points and specifying a base point and second point for moving the TIN points.

Point Elevation
Modifies the elevation of a single or multiple TIN points in one step.

Multiple
Requires a polygonal selection of multiple TIN points. Then determine the new absolute elevation or delta elevation (elevation difference).

Add Line
Adds new TIN Edge between existing TIN points.

Delete Line
Deletes TIN Edges inside the specified selection area.

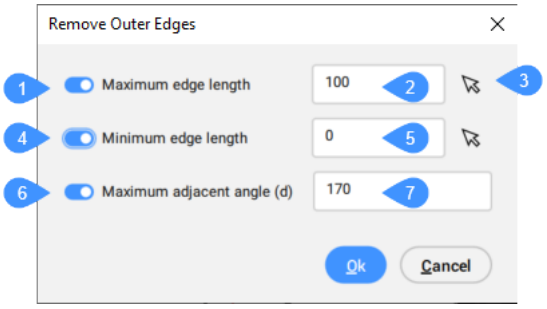
Note: The edges inside the selected area are colored red.

Remove Outer Edges
Removes triangles on the outer boundary of the TIN Surface, that have:

- greater edge length.
- shorter edge length.
- greater angle than specified.
- You can specify which of these criteria is considered when removing the outer edges.

  This method begins removing triangles from the outer boundary towards the inner part of the TIN Surface. It stops removing triangles, when the last outer triangle meets the specified criteria. That means that this method does not remove inside triangles.

  A dialog box opens:



1 Maximum edge length

2 Maximum edge length value

3 Pick points

4 Minimum edge length

5 Minimum edge length value

6 Maximum adjacent angle (d)

7 Maximum adjacent angle value (in degrees)

Maximum edge length (1)
Toggles the use of the maximum edge length.

Maximum edge length value (2)
Sets the value for maximum edge length.

Note: Triangles with an edge exceeding the specified maximum edge length are deleted from the TIN Surface.

Pick points (3)
Allows you to set the value for maximum or minimum edge length by picking points in the drawing area.

Minimum edge length (4)
Toggles the use of the minimum edge length.

Minimum edge length value (5)
Sets the value for minimum edge length.

Note: Triangles with an edge exceeding the specified minimum edge length are deleted from the TIN Surface.

Maximum adjacent angle (d) (6)
Toggles the use of the maximum adjacent angle.

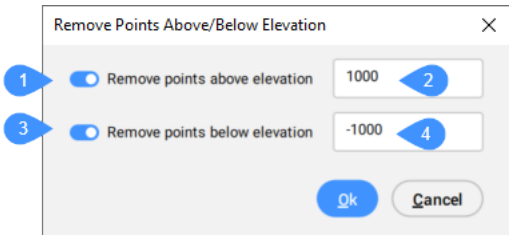
Maximum adjacent angle value (in degrees) (7)
Sets the value for maximum adjacent angle.

Note: Triangles with an inner angle exceeding a specified max. angle are deleted from the TIN Surface.

Minimize Flat Areas
Finds flat triangles and then swaps the adjacent TIN edges of those triangles so that their slope is non-zero.
This method is most useful when the TIN Surface is created from Contours. It starts at the concave side of the contours and swaps the edge of the first triangle so that its slope becomes non-zero. The process continues until the method still ensure that there are no flat triangles by swapping their edges.

Note: Minimize flat areas method swipes only the TIN edges that do not lie on breaklines.

Remove Elevations
Removes TIN points below/above the specified elevation and displays the number of points removed.
A dialog box opens:



1 Remove points above elevation

2 Elevation greater than

3 Remove points below elevation

4 Elevation less than

Remove points above elevation (1)
Toggles the option to remove TIN points above a specified elevation.

Elevation greater than (2)
Sets the elevation above which TIN points are removed.

Remove points below elevation (3)
Toggles the option to remove TIN points below a specified elevation.

Elevation less than (4)
Sets the elevation below which TIN points are removed.

SMoothing
Opens the **Smooth Surface** dialog box that allows you to smooth a TIN surface by adding additional points.

25.32 -TINEXPORT command

Exports a TIN surface via Command line.

Shape Lite Pro Mechanical BIM

25.32.1 Description

Exports a TIN surface to a specified point file format file.

25.32.2 Method

Select a TIN surface and then specify a point file format name.

25.32.3 Options within the command

Name
Indicates the TIN surface name that will be exported.

? to list tin surfaces
Lists all TIN surface names in the Command line.

Point file format name
Indicates the point file format name used for export.

? to list point file formats
Lists the names of the defined point file formats in the Command line.

25.33 TINEXPORT command

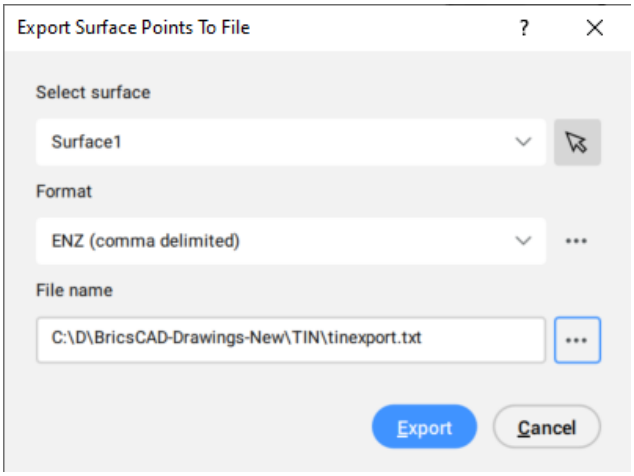
Exports a TIN surface.

Shape Lite Pro Mechanical BIM

Icon:

25.33.1 Description

Opens the **Export Surface Points To File** dialog box to choose which surface want to export.



25.33.2 Select surface

Allows you to select a TIN surface available in the drop-down list or pick a TIN surface from the drawing.

25.33.3 Format

Allows you to select a file format from the drop-down list or configure a new file format in the **Manage Surface File Formats** dialog box, which opens when you press the three dots from the right side.

25.33.4 File name

Allows you to specify the file's name and the folder where you want the TIN surface to be exported.

25.34 -TINEXTRACT command

Extracts meshes, solids, points, faces, contours, borders or intersection lines from a TIN surface, Grading, or TIN Volume Surface, via Command line.

Shape Lite Pro Mechanical BIM

25.34.1 Description

Creates a mesh or 3D solid between two TIN Surfaces / Gradings / TIN Volume Surfaces or between a TIN Surface / Grading / TIN Volume Surface and elevation or vertical offset.

25.34.2 Method

Select a surface (TIN Surface / Grading / TIN Volume Surface) and choose which entity want to extract or create.

25.34.3 Options within the command

Mesh
Creates a mesh as an offset, between surfaces or as an elevation.

vertical offset
Creates a mesh as a vertical offset above a TIN Surface / Grading / TIN Volume Surface.

between Surfaces
Creates a mesh between two surfaces (TIN Surface / Grading / TIN Volume Surface).

Elevation
Creates a mesh vertically from a TIN Surface / Grading / TIN Volume Surface to a fixed elevation height.

Separate cut and fill
If **YES**, it creates separate meshes for cut and fill.

Solid
Creates a solid as an offset between surfaces or as an elevation.

vertical offset
Creates a solid as a vertical offset above a TIN Surface / Grading / TIN Volume Surface.

between Surfaces
Creates a solid between two surfaces (TIN Surface / Grading / TIN Volume Surface).

Elevation
Creates a solid vertically from a TIN Surface / Grading / TIN Volume Surface to a fixed elevation height.

Separate cut and fill
If **YES**, it creates separate solids for cut and fill.

Points
Extracts all points on the surface.

Faces
Extracts all triangular faces of the surface.

Contours
Extracts the contours of the surfaces for an elevation.

Minor
Extracts the contours of the surfaces for the elevation value.

maJor
Extracts the contours of the surfaces for the elevation value.

All
Creates contours for all elevations.

Border
Extracts the outer boundary of the surface.

Intersection lines
Extracts 3D Polylines at the intersection between two surfaces (TIN Surfaces, Gradings, or TIN Volume Surfaces).

TIN Surface
Extracts a TIN Surface from a Grading or a TIN Volume Surface.

25.35 TINEXTRACT command

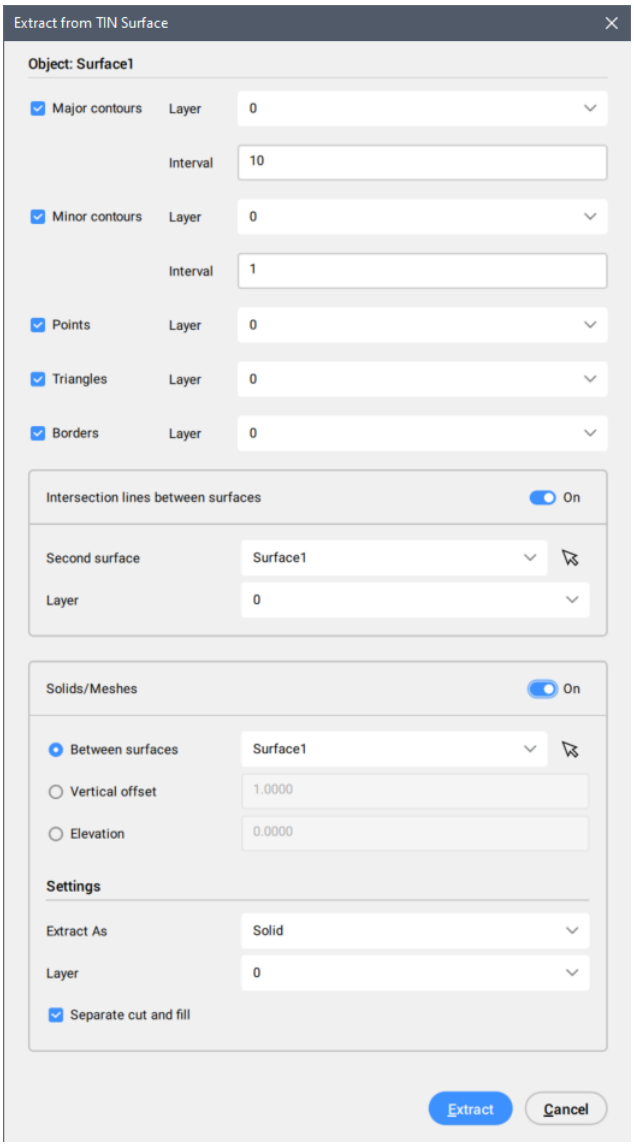
Extracts meshes, solids, points, faces, contours, borders or intersection lines from TIN surfaces, Gradings, or Volume Surfaces.

Shape Lite Pro Mechanical BIM

Icon: 

25.35.1 Description

Opens the **Extract from** dialog box to choose which entity you want to extract from a TIN surface, a Grading, or Volume Surface.



25.35.2 Major contours

Extracts the contours of a TIN surface, Grading, or TIN Volume Surface for an elevation according to major contours interval set.

Layer
Sets the layer where the major contours will be placed.

Interval
Defines the major contours interval.

25.35.3 Minor contours

Extracts the contours of a TIN surface, Grading, or TIN Volume Surface for an elevation according to minor contours interval set.

Layer
Sets the layer where the minor contours will be placed.

Interval
Defines the minor contours interval.

25.35.4 Points

Extracts all points on the surface.

Layer
Sets the layer where the points will be placed.

25.35.5 Triangles

Extracts all triangular faces of the surface.

Layer
Sets the layer where the triangular faces will be placed.

25.35.6 Borders

Extracts the outer boundary of the surface.

Layer
Sets the layer where the outer boundary will be placed.

25.36 TINMERGE command

Combines two or more TIN surfaces (or gradings) into a new TIN surface.

Shape Lite Pro Mechanical BIM

Icon: 

25.36.1 Description

Combines a base surface and two or more other TIN surfaces to merge with, which will replace the part of the base surface that it covers.

25.36.2 Methods

Select a base TIN surface and one or more TIN surfaces to merge with, and specify if you want to keep or delete the original surfaces.

25.36.3 Options within the command

Remove merged TIN surfaces

- **Yes**
  The original surfaces are deleted. This options results in a merged TIN surface that cannot be edited.
- **NO**
  The original surface to merge is not deleted. This option results in 2 TIN surfaces:
  - Original TIN surface to merge
  - When selecting and moving this surface, the merged TIN surface changes accordingly.

- Merged TIN surface.

25.37 TINMODIFY command

Deforms or smoothens a selected TIN surface.

Shape Lite Pro Mechanical BIM

Icon: 

25.37.1 Description

Modifies a selected TIN surface by deforming or smoothening it or creates a new modified TIN surface.

25.37.2 Options within the command

Deform
Modifies the selected TIN surface or creates the deformable part as a new TIN surface.

specify Contour
Creates a deformation with a contour.

select Entity
Creates a deformation with an elevated chosen closed entity.

Note: Net volume = fill-cut.

draw Polygon
Creates a deformation with an elevated polygon.

Smoothen
Modifies the selected TIN surface by smoothening it between the indicated boundaries.
Creates a circular smoothening boundary by indicating a radius value and a location where you want the smoothening.

select Entity
Smoothens the surface part between the boundaries of an entity.
Select a closed entity for the smoothening and choose a location where you want the smoothening.

draw Polygon
Smoothens the surface part between the boundaries of a polygon.
Pick points for a polygon and press ENTER to Close the polygon and choose a location where you want the smoothening.

25.38 TINPROJECT command

Projects a point-based entities or linear entities to a TIN surface.

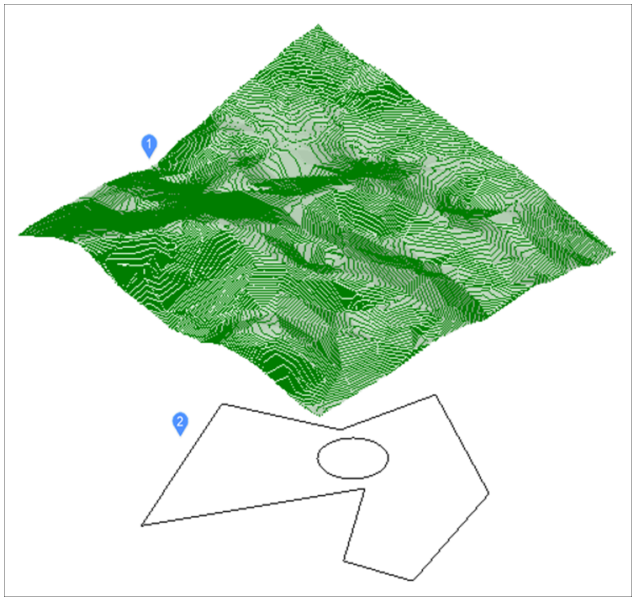
Shape Lite Pro Mechanical BIM

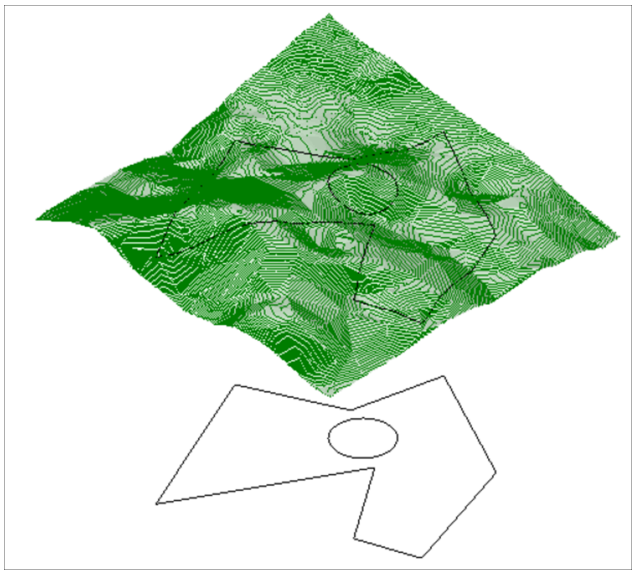
Icon: 

25.38.1 Description

Projects point entities (Point, Block, Text) or linear entities (Line, Polyline, Circle) to a TIN Surface.

Select the TIN surface (1), select the entities to project (2) and choose to keep or delete the projected entities.





25.39 TINSERT command

Inserts blocks into the cell of a table.

Shape Lite Pro Mechanical BIM

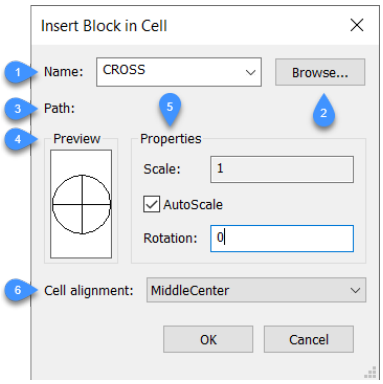
25.39.1 Method

Pick a table cell to open the **Insert Block in Cell** dialog box.

The **Insert Block in Cell** dialog box allows you to insert blocks into the cell of a table in the current drawing. You can also insert DWG and DXF files as blocks.

Note: You can have both text and blocks in a single cell.

Note: If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through the **Edit Attributes** dialog box, depending on the value of the ATTDIA (attribute dialog) variable.



1 Name

2 Browse

3 Path

4 Preview

5 Properties

6 Cell alignment

25.39.2 Name

Specifies the name of the block, whose definition exists in the drawing. It can also be a DWG or DXF file on the computer or network.

25.39.3 Browse

Selects a DWG or DXF file from your computer or network. Opens the **Open drawing file** dialog box.

Note: Placing a large drawing in a small cell can lead to program instability.

25.39.4 Path

Displays the path to the block, if the block was opened from a DWG or DXF file.

25.39.5 Preview

Displays a preview of the selected block definition.

Note: When the cell is small, the block or drawing may be barely perceptible. You can increase the size of the row or column to accommodate the blocks. A cell can hold text and one or more blocks. When a 3D model is pasted into the cell, the 2D plan view is displayed.

25.39.6 Properties

Defines additional properties of the selected block.

Scale

Specifies the scaling value of the block:

- Values larger than 1 enlarge the block.
- 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.

Note: This option is unavailable when AutoScale is turned on.

AutoScale

Sizes the block to fit the cell:

- **On**: the block is resized to fit the cell.
- **Off**: the cell is resized to fit the block.

Rotation

Rotates the block around its center point (and not its insertion point). Enter an angle:

- Positive numbers rotate the block counter clockwise.
- 0 - does not rotate the block.
- Negative numbers rotate the block clockwise.

Note: The specified rotation angle depends on the AUNITS system variable.

25.39.7 Cell alignment

Aligns the block in the cell.

25.40 TINVOLUME command

Creates a TIN volume surface between a base and comparison TIN surfaces or an elevation.

Shape Lite Pro Mechanical BIM

Icon: 

25.40.1 Method

When the TIN Volume Surface associativity option of the Civil / Associativity user preference is set, TIN volume surfaces automatically rebuild when their source surfaces change.

Select the base and the comparison TIN surfaces between which a TIN volume surface will be created and select a bounding area for the TIN volume surface.

25.40.2 Options within the command

Elevation

Creates a TIN volume surface between a base and an elevation.
Select the base TIN surface, enter the desired elevation value and select a bounding area for the TIN volume surface.

Note: In the **Properties** panel of a TIN Volume Surface there are **TIN Volume Statistics** properties:

- **Cut volume**: reports the cut volume.
- **Fill volume**: reports the fill volume.
- **Cut 2D area**: reports the 2D area of the cut portions of the TIN Volume Surface.
- **Fill 2D area**: reports the 2D area of the fill portions of the TIN Volume Surface.
- **Cut 3D area**: reports the 3D area of the cut portions of the TIN Volume Surface.
- **Fill 3D area**: reports the 3D area of the fill portions of the TIN Volume Surface.

25.41 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 left mouse button.

Shape Lite Pro Mechanical BIM

Icon: 

25.41.1 Method

Select the TIN surfaces, move the cursor over the surface and a line representing the drop path is shown in real time. Click the left mouse button to create the 3D polyline of the water drop path.

25.42 TJUST command (Express Tools)

Changes the justification point of text, mtext, and attribute definition entities.

Shape Lite Pro Mechanical BIM

Icon: 

25.42.1 Options within the command

Start
Justifies the text at the left of the baseline.

Center
Justifies the text in the center of the baseline.

Middle
Justifies the text in the center.

Right
Justifies the text on the right of the baseline.

TL
Justifies the text at the top left of the text.

TC
Justifies the text at the top in the middle.

TR
Justifies the text at the top right of the text.

ML
Justifies the text in the middle left.

MC
Justifies the text in the middle center.

MR
Justifies the text in the middle right.

BL
Justifies the text at the bottom left.

BC
Justifies the text at the bottom center.

BR
Justifies the text at the bottom right.

25.43 TOLERANCE command

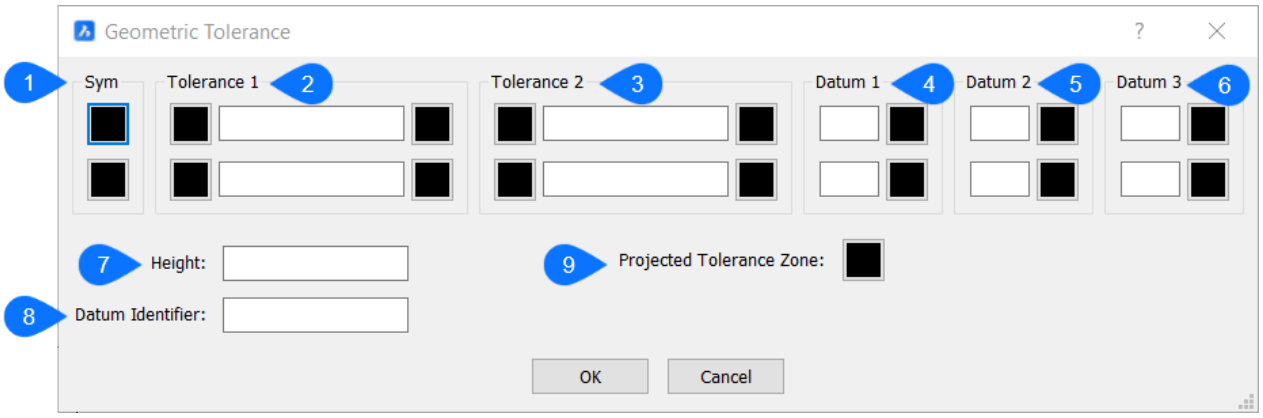
Opens the **Geometric tolerance** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

25.43.1 Description

Opens the **Geometric tolerance** dialog box to add tolerance symbols to the current drawing.



1 Symbol

2 Tolerance 1

3 Tolerance 2

4 Datum 1

5 Datum 2

6 Datum 3

25.43.2 Symbol

Specifies a tolerance symbol through the **Symbol** dialog box.

25.43.3 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 through the **Material condition** dialog box.

25.43.4 Datum 1, 2 & 3

Specifies the datum reference (value and material condition).

Value
Specifies the datum value.

Material condition
Specifies the material condition through 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

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

Material Condition Symbols

Symbol	Definition
	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.
	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

25.44 TOOLBAR command

Toggles the display of toolbars at the Command line.

Shape Lite Pro Mechanical BIM

25.44.1 Method

Enter the name of the toolbar or choose the All option to turn all toolbars on or off.

25.44.2 Options within the command

Show
Displays the toolbar(s).

Hide
Hides the toolbar(s).

Left
Docks the toolbar(s) on the left.

Right
Docks the toolbar(s) on the right.

Top
Docks the toolbar(s) to the top.

Bottom
Docks the toolbar(s) to the bottom.

Float
Show the floating toolbar(s).

25.45 -TOOLBAR command

Toggles the display of toolbars at Command line.

Shape Lite Pro Mechanical BIM

25.45.1 Description

For more information see TOOLBAR command.

25.46 TOOLPALETTES command

Opens the **Tool Palettes** panel.

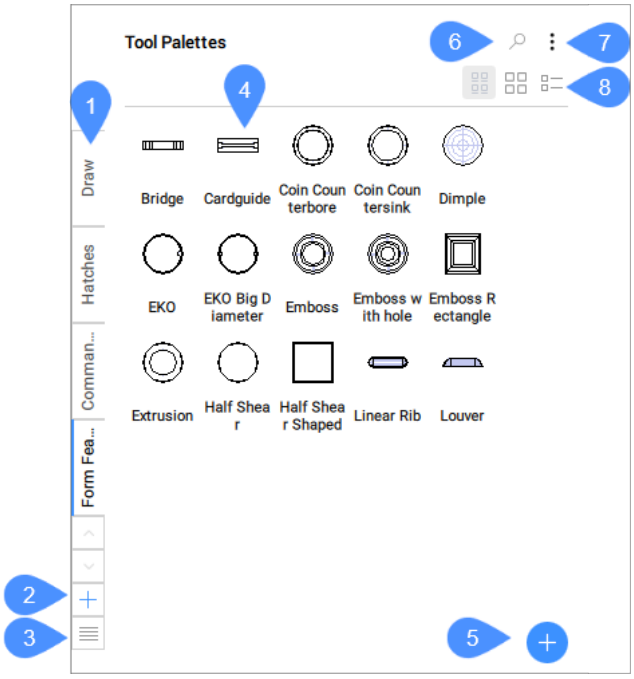
Shape Lite Pro Mechanical BIM

25.46.1 Description

Opens the **Tool Palettes** panel to display it in the current workspace. The **Tool Palettes** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Tool Palettes** panel can be either floating, docked or stacked.

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.



1 Palettes Tabs

2 Add tool palette button

3 Tool Palettes list button

4 Tools

5 Add button

6 Search

7 Tool palettes context menu

8 View modes

25.46.2 Palettes Tabs

Displays a list of available palettes.

Note: The current palette is highlighted with a blue line.

Note: The tab items have a fixed height, so not always the name is legible. The entire name of the palette will be displayed when hovering over the tab. A palette can be renamed by double clicking on it, or from the context menu which is displayed by right clicking the panel tab.

Note: Using the up and down arrow button the user can see the hidden tabs.

- Tab labels should be "Elided"

Note: The Tab bar changes his position depending on the docking of the **Tool Palettes** panel.

25.46.3 Add tool palette button

Adds a new tool palette.

25.46.4 Tool Palettes list button

Lists all available tool palettes. User may choose the current palette.

25.46.5 Tools

Displays a list of available tools in the current palette.

Right-click menus offer additional functionality for tools:

Note: Rectangle selection is available to select multiple tools.

Cut
Removes the selected tools from the palette.

Copy
Copies the selected tools to the clipboard.

Delete
Removes the selected tools from the palette.

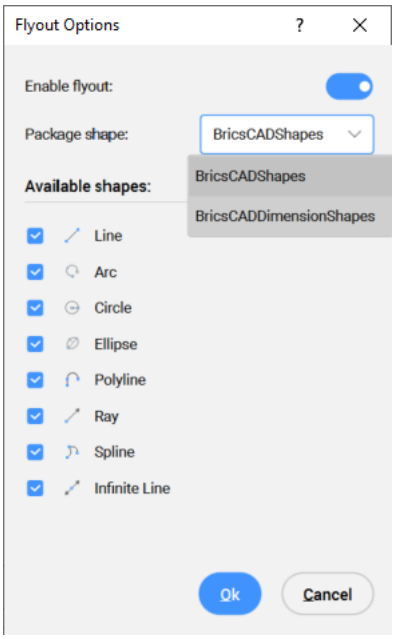
Rename
Renames a tool.

Specify Image…
Allows you to specify another image for the tool.

Note: To restore the original tool image, right-click the tool and choose **Remove Image...** in the context menu.

Manage Flyout options...
Opens the **Flyout Options** dialog box which allows you to enable tool flyouts.

Note: Tool flyouts via XTP import are supported.



The predefined flyouts Package Shape are available in: C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\shapes.btc file.

Properties…
Opens the **Tool Properties** dialog box, which allows you to edit a tool (see **Tool Properties dialog box** article).

25.46.6 Add button

Allows you to add new **tools**, **texts** and **separators**.

Note:

- Texts and separators can be cut, copy and pasted from the context menu.
- Texts and separators can be drag & dropped.

25.46.7 Search

Allows you to search for specific tools in the current palette. The list is shortened to display only the tools that contain the entered sequence of characters, regardless of where they are located in the tool's name.

25.46.8 Tool Palettes context menu

Opens the tool palettes context menu.

View Options…
Opens the **View options** dialog box, which allows you to modify the display of the tool icons (see **View options dialog box** article).

Add text
Adds a text into the current palette.

Add Separator
Adds a separator into the current palette.

Update Palette
Updates the palettes.

New Palette
Allows you to add a new palette.

Delete Palette
Allows you to delete the current palette.

Rename Palette
Allows you to rename the current palette.

Customize Palettes…
Opens the **Customize** (Tool Palettes) 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** (Tool Palettes) dialog box, which allows you to add a new tool (see **Customize (Tool Palettes) dialog box** article).

25.46.9 View modes

Icon with text
Displays tools in a grid view in icon and text mode.

Icon only
Displays tools in a grid view in icon mode only.

List view
Displays the tools in a list view in the current palette.

25.47 TOOLPALETTESCLOSE command

Closes the **Tool palettes** panel.

Shape Lite Pro Mechanical BIM

25.47.1 Description

Closes the **Tool Palettes** panel to hide it from the current workspace. If the **Tool palettes** panel is stacked when you close it, the Tool palettes tab or icon is removed from the stack.

25.48 -TOOLPANEL command

Toggles the display of panels.

Shape Lite Pro Mechanical BIM

25.48.1 Description

Turns the display of panels on and off, such as the **Properties** and **Tips** panels.

25.48.2 Method

There are two methods to toggle panels:

- Enter the name of the panel, and then choose a display option.
- Enter ? to list the names of all panels in the program.

25.48.3 Options within the command

Show
Displays the tool panel.

Hide
Hides the tool panel.

Toggle
Toggles the display of the tool panel On and Off.

25.49 TORIENT command (Express Tools)

Rotates text, mtext, attribute definitions and block attribute entities to new orientation.

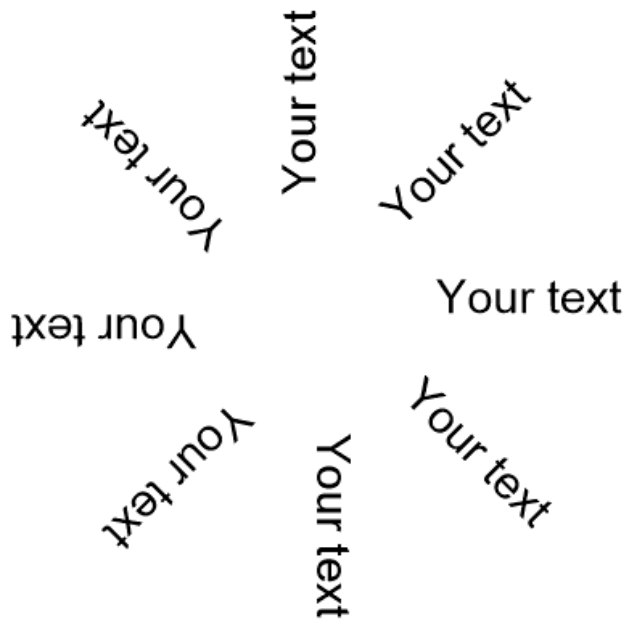
Shape Lite Pro Mechanical BIM

Icon:

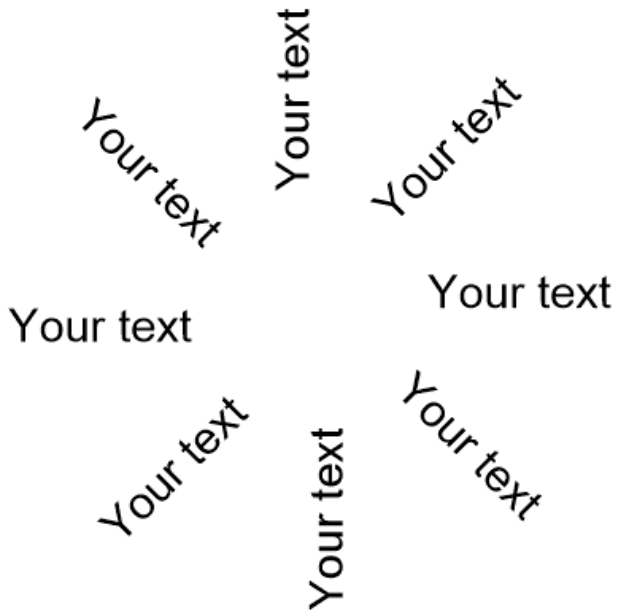
25.49.1 Method

Select entities and specify the rotation angle. The entities are rotated around their middle point in 180-degree increments.

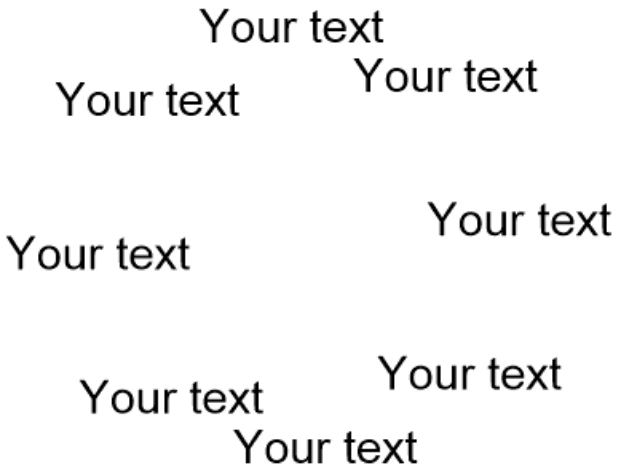
- **Original text**:



- **most Readable**:



- **Specified angle of 0 degree**:



25.49.2 Options within the command

most Readable
Orients the text in the most readable way.

Align with line
Aligns a text along a line or a polyline.

25.50 TORUS command

Creates a 3D Solid in the shape of a torus.

Shape Lite Pro Mechanical BIM

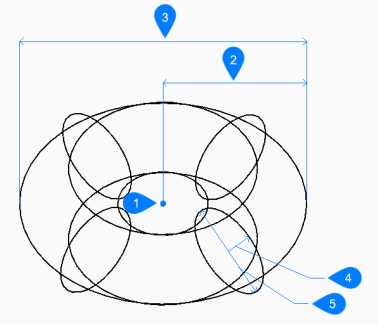
Icon:

Alias: TOR

Note: In BricsCAD Lite, which doesn't support 3D Solids, the TORUS command launches the AI_TORUS command.

25.50.1 Description

Creates a 3D Solid in the shape of a torus. Choose from a combination of options including center and the radius or diameter of both the whole torus and the body of the torus.



1 Center

2 Radius of the whole torus

3 Diameter of the whole torus

4 Radius of the body

5 Diameter of the body

25.50.2 Options within the TORUS command

Center of whole torus
Specifies the center of the torus.

Set radius of whole torus
Specifies the overall radius of the torus. The radius is measured from the center of the whole torus to the center of the body (tube) of the torus.

Diameter
Specifies the diameter of the entire torus. The diameter is twice the distance from the center of the whole torus to the center of the body (tube) of the torus.

Set radius of body of torus
Specifies the radius of the body (tube) of the torus.

Diameter
Specifies the diameter of the body of the torus.

25.51 TPNAVIGATE command

Loads tool palettes at the Command line.

Shape Lite Pro Mechanical BIM

25.51.1 Description

Loads a tool palette or palette group by name. If the Tool Palettes panel is not already open, then it is displayed (short for "tool palette navigation"). This command is meant for use by macros.

25.51.2 Method

There are two methods to load tool palettes:

- Specify the tool palette to display by entering the name of a palette.
- Specify the palette group to display by entering the name of a group.

25.52 TRACE command

Draws traces.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command is rarely used as the PLINE command is more convenient.

25.52.1 Description

Draws traces, like wide lines, with mitered vertices.

The trace is not drawn until the second pick point. The delay allows BricsCAD to determine the mitering needed at vertices, which this command does automatically.

Note: The FILLMODE system variable affects the look of traces.

25.53 TRANSPARENCY command

Toggles the transparency of monotone images.

Shape Lite Pro Mechanical BIM

Icon:



25.53.1 Method

This command allows to set the transparency for a certain type of images. When transparency is on, the background color of the image is transparent.

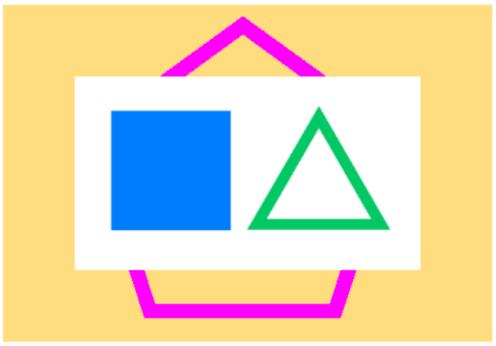
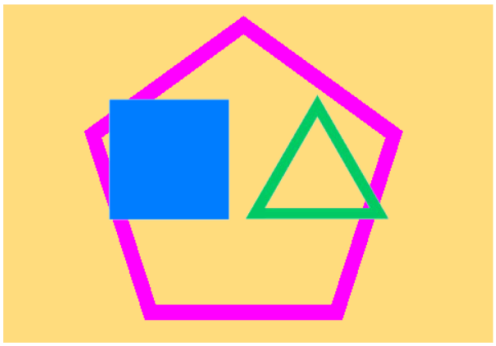
Note: To change the transparency of entities, use the Transparency property in the LAYER and PROPERTIES commands.

There are two main categories of images that can be processed by this command:

- Images that contain transparency in form of an alpha channel, usually PNG and TIFF formats.
- Images that do not contain an alpha channel.

Example of an image that contains an alpha channel

The images with an alpha channel can be processed by the TRANSPARENCY command. Before running the command on the image, its transparent pixels will be displayed as white. After the command has been run on the image, its transparent pixels will no longer be displayed.

	When the image with an alpha channel is placed on top of other entities, the transparent pixels are displayed as white.
	After running the command on the image, the transparent pixels are no longer displayed.

Example of an image that does not contain an alpha channel

Before processing this kind of images, they must be converted to a monochrome palette using some photo editor.

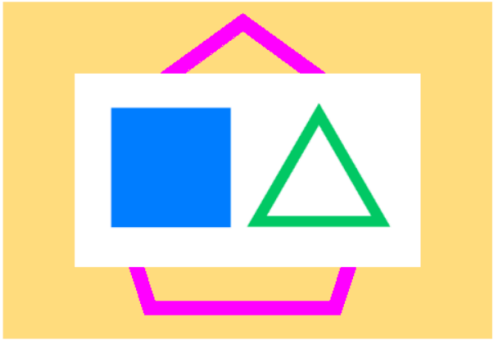
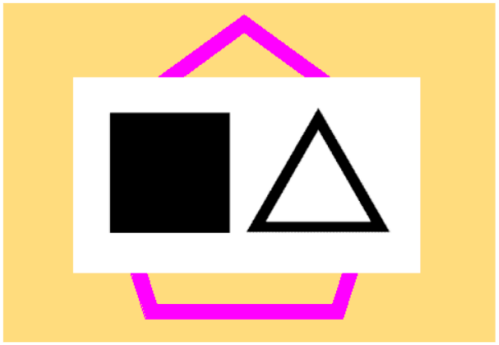
There are some conditions for an image to be affected by this command and to obtain a satisfactory result:

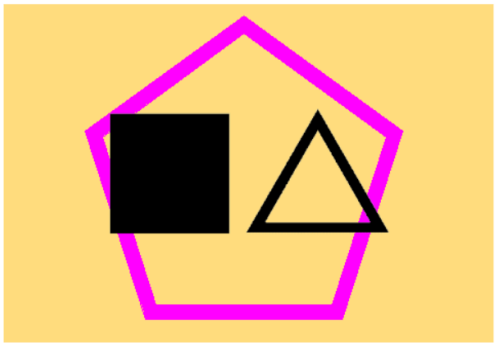
1. The image must be in TIFF, BMP or PNG graphical format.
2. The background color of the image must be white or very close to white.
3. The image must have a good contrast between the objects in focus and the background.
4. In a photo editor, there should be applied a Threshold adjustment. This way, there would be only two colors in the image: black and white.

5. The image must be saved in a graphical format with a color bit depth of 1.

Note: Saving with a color bit depth of 1 is an important step. Even if there are only black and white colors in the image, the graphical format could be different than this. Check the graphical format before attaching the image to the drawing.

After running the command on the image, the white pixels will become transparent.

	The image with white background placed on top of other entities looks like this. This is just for visualizing purposes. The original image should not be attached to the drawing.
	The image has been applied to a Threshold adjustment and converted to a color bit depth of 1, and then attached to the drawing.

	After running the command on the image, the transparent pixels are no longer displayed.

Other graphical formats, as JPEG, are not affected by this command. The files that have one of these formats must be converted using a photo editor that can output according to the specifications above.

25.53.2 Options within the command

selection options(?)
Allows to choose a selection method. See the SELECT command.

Enter transparency mode [ON/Off]
Toggles the transparency of the selected images.

25.54 TREX command (Express Tools)

Extends or trims entities.

Shape Lite Pro Mechanical BIM

25.54.1 Method

Select cutting/boundary edges or press Enter for all entities.

Note: Click entities to trim or hold the Shift key + click to extend.

See also the EXTEND and TRIM command articles.

25.55 TRIM command

Trims entities by a cutting line.

Shape Lite Pro Mechanical BIM

Icon:

Alias: TR

25.55.1 Method

There are two methods:

- Trims entities.
- Extends entities while holding down the Shift key.

Note:

- 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.
- Selected circles, lines, or polylines consisting of only one segment entity that does not cross any other entities will be deleted.

25.55.2 Options within the command

mOde
Defines the mode in which the command is acting.

Note: The TRIMEXTENDMODE system variable also controls the trim mode.

Standard
Acts in standard mode, where you must first specify the cutting and boundary edges.

Quick
Acts in quick mode, where all entities automatically act as cutting edges.

Note: In **Quick** mode, the TRIMEDGES system variable determines whether the edges of the hatches or the hatch patterns with their inside entities count when trimming and extending.

cuTting edges
Defines the cutting entities used as a boundary for trim.

Fence
Trims all entities that cross the selection fence. The selection fence is a series of temporary line segments. The selection fence does not form a closed loop.

Crossing
Trims entities within and crossing a rectangular area defined by two points.

Edge
Toggles between **Extend** and **No extend**.

Note: The EDGEMODE system variable also controls how cutting edges are checked.

Extend
Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.

No extend
Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.

Projection
Specifies the projection method used when extending objects.

Note: The PROJMODE system variable also controls the projection mode.

No projection
Extends only those entities that intersect real boundaries in 3D space.

xy plane of Ucs
Projects entities and boundaries to the XY plane of the current UCS, and then extends the projected entities that would intersect projected boundaries.

current View
Projects entities into the current view, and then extends them accordingly.

eRase
Deletes the selected entity.

Undo
Undoes the last trim action.

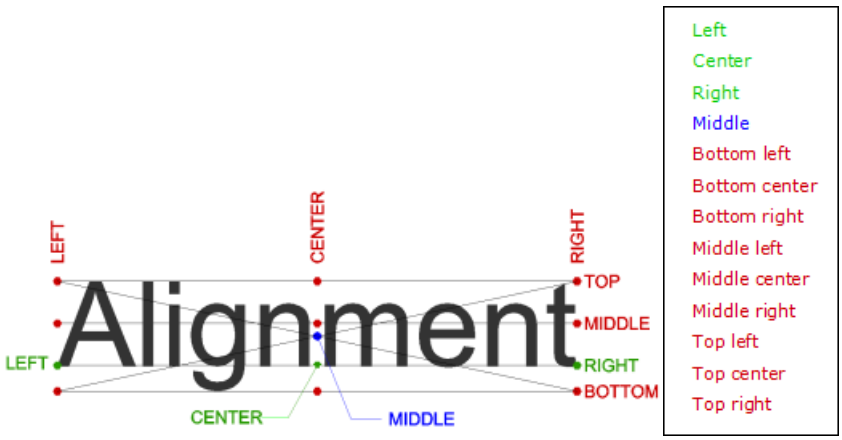
25.56 TSCALE command (Express Tools)

Scales texts, mtexts, attributes and attribute definitions.

Shape Lite Pro Mechanical BIM

25.56.1 Method

Select the entities and specify the justification used for scaling.



25.56.2 Options within the command

Scale
Allows you to specify a scale factor.

Height
Allows you to specify the text height.

25.57 TSPACEINVADERS command (Express Tools)

Creates a selection set of the text entities that have another entity that overlaps it.

Shape Lite Pro Mechanical BIM

25.57.1 Method

Select the entities you want to check for overlapping entities or type **all** to check the entire drawing. The number of overlapping entities is displayed in the command line.

25.57.2 Options within the command

Should each text be queried individually?
Allows you to decide if each text should be queried individually.

Yes
All overlapping entities are highlighted one by one. It allows you to choose if you want to add the highlighted entity to the selection set.

No
The command ends and creates a selection set.

25.58 TUTORIALSOPEN command

Provides you direct access to in-product tutorials.

Shape Lite Pro Mechanical BIM

Icon: 

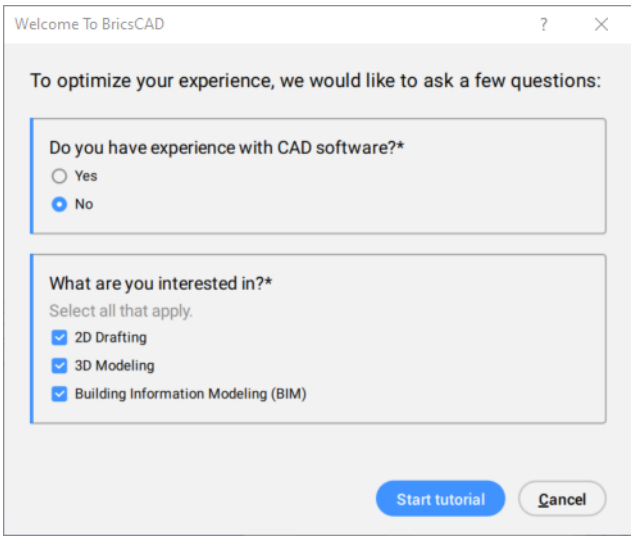
25.58.1 Description

Launches in-product tutorials that teach you the basics of BricsCAD or get to know some of BricsCAD's unique features, with step-by-step instructions, animated images, and tailored sample drawings, designed to help you practice each new command.

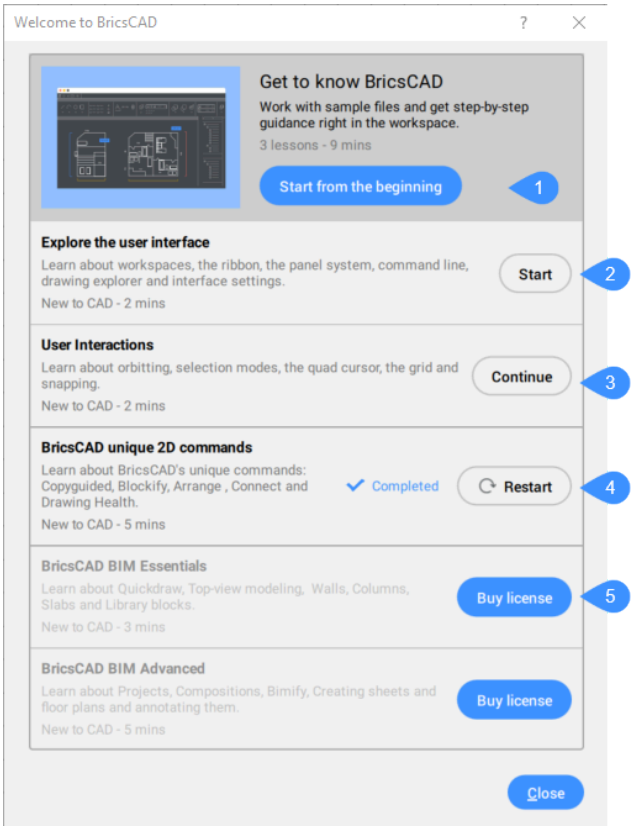
Note: An internet connection is required.

25.58.2 Method

Opens the **Welcome To BricsCAD** dialog box, where you have to answer a few questions about your experience with CAD software and your areas of interest.



Then, depending on the experience, it opens a list with lessons.



1 Start from the beginning

2 Start

3 Continue

4 Restart

5 Buy license

Start from the beginning
Starts the tutorial from the beginning.

Start
Starts specific lessons.

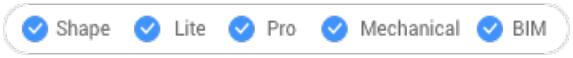
Continue
Continues the lessons that have already been started.

Restart
Starts lessons that have already been completed.

Buy license
Opens the Bricsys store page to purchase a higher license level. That will give you access to lessons that require a higher license level.

25.59 TXT2MTXT command (Express Tools)

Combines a selection of TEXT and MTEXT entities into a single MTEXT entity.



Alias: COMBINETEXT

25.59.1 Method

Select the TEXT and MTEXT objects to combine. The combination is made depending on the value of the COMBINETEXTMODE system variable.

25.59.2 Options within the command

SEttings
Displays the options of the COMBINETEXTMODE system variable in a dialog box.

Combine into a single mtext
Combines the selected TEXT entities into a single MTEXT entity.

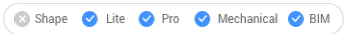
Sort top-down
Specifies the order of the selected text entities by descending vertical position.

Word-wrap text
Combines all selected TEXT entities into a single line and then wraps any text that exceeds the width of the MTEXT to the next line. The MTEXT width matches the width of the largest text entity in the selection. Paragraphs in the selected MTEXT entities are preserved.

Uniform linespacing
Applies consistent interline spacing.

25.60 TXTEXP command (Express Tools)

Explodes text into polylines.



25.60.1 Method

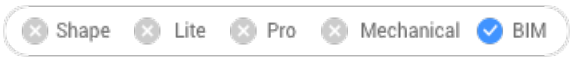
Select the TEXT entities to be exploded.

Note: SHP and TTF fonts are exploded to individual polylines and polyarcs.

Note: The command does not explode attributes in blocks or text in tables, or text in another space (Model vs Paper). Standalone attributes are, however, exploded.

25.61 TYPEDPLANS command

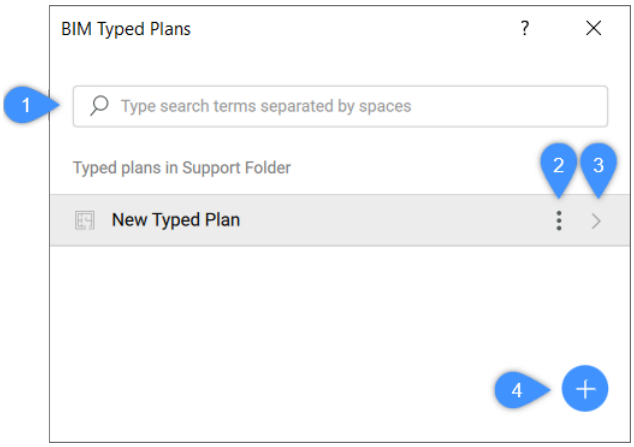
Opens the **BIM Typed Plans** dialog box.



Icon:

25.61.1 Description

Opens the **BIM Typed Plans** dialog box, listing all Typed Plans in the Support folder and allowing you to edit them.



1 Search

2 Context menu

3 Edit typed plan

4 Add typed plan

25.61.2 Search

Searches all the Typed Plans that matches the words you enter in the search box.

25.61.3 Context menu

Clone
Creates a copy of the selected Typed Plan.

Remove
Removes the selected Typed Plan.

Rename
Edits the name of the selected typed plan. Press Enter to save and apply the changes.

25.61.4 Edit typed plan

Opens the **BIM Typed Plan Editor** dialog box which allows you to edit the selected Typed Plan.

Note: You can also open the **BIM Typed Plan Editor** for any Typed Plan by clicking on it's name in the list.

25.61.5 Add typed plan

Creates a new Typed Plan.

You can create a template by clicking on the button. By default, the Typed Plan is named **New Typed Plan** with the text highlighted. After you have added a relevant name for the Typed Plan, press **Enter** to save and apply the changes. The inserted Typed Plans will be sorted alphabetically. Use the scroll bar to navigate through the list.

26. U

26.1 U command

Reverses the action of the previous command.

Shape Lite Pro Mechanical BIM

Icon:

26.1.1 Description

Reverses the action of the previous command to restore entities to their prior state.

This command displays no prompts and has no options.

26.2 UCS command

Creates and displays user-defined coordinate systems (UCS) through the Command line.

Shape Lite Pro Mechanical BIM

Icon:

26.2.1 Method

Specify a new UCS origin through one, two or three points in the drawing or choose one of the command's options.

26.2.2 Options within the command

Face
Aligns the UCS to a face of a 3D solid.

Note: This option works only with flat 3D faces and not curved ones, such as on a sphere.

Flip
Reverses the direction of the Z-axis.

Xrotate
Rotates the UCS 180˚about the X-axis.

Yrotate
Rotates the UCS 180˚about the Y-axis.

NAmed
Creates, restores and deletes UCS names.

Restore
Allows you to enter a UCS name to restore.

Save
Allows you to enter a name for saving the UCS.

Delete
Allows you to enter the UCS name to delete.

?
Lists the names of UCSs in the drawing.

Note: Type:

- *to list all UCS names.
- name* to list name of UCSs that begin with name.
- name to list the UCS specified by name.

Entity
Aligns the UCS to a selected entity.

Select entity for UCS definition
Represents the orientation and type of entity defining the orientation of the UCS.

Note: The UCS origin is located to the end point, center point, or vertex closest to the pick point. The X-axis is aligned with the entity or an edge. The X,Y-plane is aligned to the entity's plane. For ambiguous objects with obvious orientation, like a circle, the orientation is maintained.

POintcloud
Defines the UCS based on the normal of a point selected in a point cloud.

Note: Make sure that in the **3d entity snap mode** category of the **Settings** dialog box the **Turn off all 3D snaps**option is disabled and the **Point cloud nearest point** option is enabled (see the **3DOSMODE system variable** article).

Select point for UCS definition
Defines the origin and gets the Z direction from the point cloud normal. Accept or continue to define the X-axis.

Point on X-axis
Allows you to select a point in the point cloud to define the X-axis. Accept or continue to define the Y-axis.

Point on the XY-plane with positive Y value
Allows you to select a point in the point cloud to define the XY-plane.

Note: The **Flip**, **Xrotate**, and **Yrotate** options become available after accepting any of the options above.

Previous
Changes the UCS to the previous one.

View
Sets the UCS to the current viewpoint.

Note: The x axis and y axis are parallel to the view edges. The z axis is perpendicular to the view, with positive Z-axis pointing to the viewer. The origin is copied from the previous coordinate system.

X
Rotates the current UCS about the X-axis.

Y
Rotates the current UCS about the Y-axis.

Z
Rotates the current UCS about the Z-axis.

Z Axis
Sets the UCS relative to the Z-axis.

Note: Y-plane is perpendicular to the Z-axis with the X-axis horizontal and the Y-axis pointing upwards.

Move
Moves the origin of the current UCS, keeping the orientation of the axes.

World
Switches to the world coordinate system (WCS).

26.3 UCSICON command

Controls the display of the UCS icon.

Shape Lite Pro Mechanical BIM

26.3.1 Description

The UCS icon is displayed differently, depending on the visual style and workspace.

The colors denote the directions of the axes. It can be controlled by the COLORX, COLORY, COLORZ system variables.

26.3.2 Options within the command

ucs icon ON
Displays the UCS icon.

ucs icon OFf
Disables the UCS icon.

display in All views
Applies the changes to all viewports.

display at ORigin
When the origin is outside of the viewport, the UCS icon is displayed at the corner defined by the UCSICONPOS system variable.

display in Corner
Displays the UCS icon in the viewport corner defined by the UCSICONPOS system variable.

Note: The values for UCSICONPOS system variable are displayed in the **Settings** dialog box.

26.4 UNDEFINE command

Removes access to commands temporarily.

Shape Lite Pro Mechanical BIM

26.4.1 Method

Enter the name of the command to undefine.

Undefined commands are accessed:

- By prefixing the command name with a dot, such as: .CommandName.
- By using the REDEFINE command to recover the command name.

Note: Commands are undefined by developers who want to replace the command with a more extensive version they're written, or to prevent neophytes from using destructive commands, like ERASE and EXPLODE.

26.5 UNDO command

Undoes one or more commands.

Shape Lite Pro Mechanical BIM

Icon: 

26.5.1 Description

Undoes one or more commands. BricsCAD reports the commands being undone.

Note: Some commands cannot be undone.

26.5.2 Options within the command

Number of steps to undo
Specifies the number of commands to undo by entering a number.

Mark
Sets a marker.

Back to mark
Undoes all commands back to when the marker was set.

BEgin set
Sets the start to a group that groups the following commands. The UNDO command treats the commands in the set as a single undo.

End set
Ends the group of commands.

Control
Specifies several options for the command.

None
Turns off the undo mechanism.

Note: This is useful when disk space is running low, because the undo mechanism uses disk space.

One
Limits this command to a single undo. The UNDO command is changed into the U command.

All
Turns on the Undo mechanism.

Layer
Specifies whether the UNDO command combines Layer dialog operations.

Auto
Considers all commands executed by a macro as a single undo.

Note: When Control is set to **None** or **One**, the **Auto**, **Begin** and **Mark** options are not available.

26.6 UNDOENT command

Undoes revisions to entities.

Shape Lite Pro Mechanical BIM

26.6.1 Description

Reverses editing changes made to individual entities. Acts like the UNDO command, but is specific to each entity.

This command treats a block, xref, or 3D solid as a single entity, but a group as individual entities.

26.6.2 Options within the command

Number of steps to undo
Specify the number of editing steps to reverse. After the initial revision is undone, the entity is erased from the drawing.

rEvisions
Report the number of revisions the selected entity has undergone.

eXit
Exit the command.

Redo
Reverses the undo action.

Note: When an entity is connected to other ones, such as an edge connected to a face, BricsCAD asks if the other entities should be reverted. Alternatively, use the History drop-down list in the Properties panel to undo revisions to entities.

26.7 UNGROUP command

Explodes groups of entities.

Shape Lite Pro Mechanical BIM

Icon: 

26.7.1 Method

There are two methods to explode a group of entities:

- By selecting the group.
- By entering its name in the Command line.

26.7.2 Options within the command

Name
Enter the name of a group to explode.

?
Lists the groups existing in the drawing.

26.8 UNION command

Performs Boolean union operations on 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icon:

Alias: UNI

Note: In the BricsCAD Lite license level, the command applies to region entities only.

26.8.1 Description

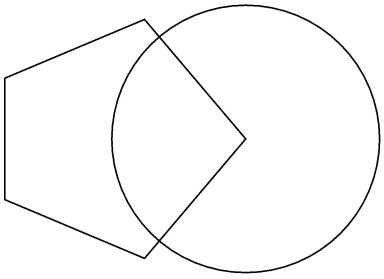
Performs Boolean union operations on 3D solids and 2D regions by adding set of entities to another set to form a single entity.

Note: The resulting entity takes on the properties of the ACIS entity selected first.

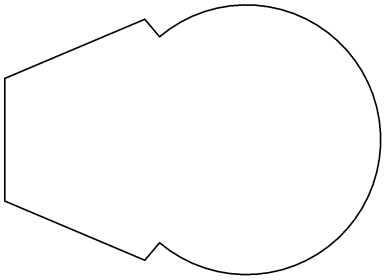
26.8.2 Method

Specify the ACIS entities to join into a single entity. The program joins regions and 3D solids into a single entity.

The entities selected:



Result:



Note: You can interrupt the command by pressing the CANCEL button.

26.9 UNISOLATEOBJECTS command

Unhides entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: UNHIDE, UNHIDEOBJECTS, UNISOLATE

26.9.1 Description

Unhides entities that were hidden by the HIDEOBJECTS and ISOLATEOBJECTS commands.

Note: The UNISOLATEOBJECTS command is enabled under BEDIT sessions.

26.10 UNITS command

Opens the **Settings** dialog box with the **Drawing Units** category expanded.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DDUNITS, UN

26.10.1 Description

Opens the **Settings** dialog box with the **Drawing Units** category expanded to view and modify relevant system variables.

26.11 -UNITS command

Sets units of linear and angular measurements in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -UN

transparent: **'-units**

26.11.1 Options wihin the command

Unit mode (LUNITS)

1 Scientific 4.225E+01

2 Decimal 42.25

3 Engineering 3'-6.25"

4 Architectural 3'-6 1/4"

5 Fractional 42 1/4

Number of decimal places for LUNITS

Specifies the display precision for decimal units. Enter a number between 0 and 8:

- **0** - 0 decimal places, such as 0.
- **1** - 1 decimal places, such as 0.0
- **2** - 2 decimal places, such as 0.00
- **3** - 3 decimal places, such as 0.000
- **4** - 4 decimal places, such as 0.0000
- **5** - 5 decimal places, such as 0.00000
- **6** - 6 decimal places, such as 0.000000
- **7** - 7 decimal places, such as 0.0000000
- **8** - 8 decimal places, such as 0.00000000

This option also affects the display precision of fractions used by architectural and fractional units:

- **0** - 0 fractional precision, such as 1
- **1** - 1/2
- **2** - 1/4
- **3** - 1/8
- **4** - 1/16
- **5** - 1/32
- **6** - 1/64
- **7** - 1/128
- **8** - 1/256

Angular unit mode (AUNITS)

Specifies the display style of angular units. Enter a number:

1 Decimal degrees 90.0

2 Degrees/minutes/seconds 90d0'0

3 Grads 100.00g

4 Radians 1.57r

5 Surveyor's units N 00d0'0"E

There are 400 grads in a circle. There are 2*pi radians (approximately 6.282) in a circle. The N and E in Surveyor's units refers to North and east.

AUNITS variables stores the style of angular units.

Number of decimal places for angular units

Specifies the number of decimal places. Enter a number between 0 and 8.

Angle 0 direction

Specifies the direction for 0 degrees. Enter an angle, or pick two points in the drawing. The default is the positive x axis. You can also rotate the drawing through the Snap command's Rotate option.

Do you want angles measured clockwise?
Specifies the direction in which to measure angles:

- **Yes** - measures angles clockwise
- **No** - measures angles counterclockwise (default)

26.12 UPDATEFIELD command

Updates the display values of fields.

Shape Lite Pro Mechanical BIM

Icon: 

26.12.1 Description

Fields are lines of text that change as conditions in drawings change.

UPDATEFIELD forces selected field text to update, to reflect the changes.

26.13 UPDATELAYOUTBLOCK command

Updates an inserted layout block reference.

Shape Lite Pro Mechanical BIM

26.13.1 Description

Updates a layout block reference inserted using the INSERTLAYOUT command according to the source drawing layout changes.

26.13.2 Options within the command

Select inserted block
Allows you to select the inserted layout block reference that should be updated.

Note: For the block to be updated, the name of the source file must be the one from when the layout was inserted.

26.14 UPDATESKETCH command

Changes the parameters of the sketch features.

Shape Lite Pro Mechanical BIM

26.14.1 Description

Allows you to modify a parameter of a sketch feature.

Note: The EXTRUDE, LOFT, SWEEP, and REVOLVE commands and their children features SUBTRACT, and UNITE, create sketch features when the CREATESKETCHFEATURE system variable is ON. They are available in the **Mechanical Browser** panel.

26.14.2 Method

Specify the feature name, then the parameter name, and set the new value for it.

26.15 URL command

Opens the default Web browser.

Shape Lite Pro Mechanical BIM

Icon:

26.15.1 Description

Opens the default Web browser to browse the internet starting from a specified URL. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

26.16 USAVE-CLOSEALL command (Express Tools)

Closes all drawings without saving them, except for the active one.

Shape Lite Pro Mechanical BIM

27. V

27.1 VBAIDE command

Opens the Microsoft Visual Basic editing window for writing and debugging VBA code (short for "visual basic for applications integrated development environment").

Shape Lite Pro Mechanical BIM

Icon:

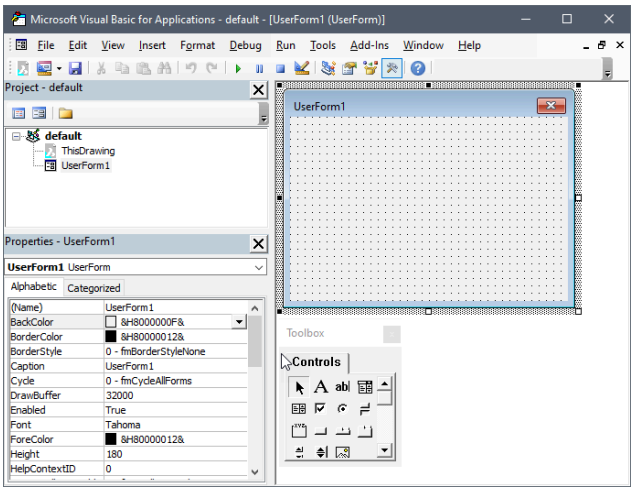
Alias: VBA

Keyboard Shortcuts: **Alt+F11**

Note: Command only available on the Windows platform.

27.1.1 Description

Displays a window:



See the help files provided by Microsoft for Visual Basic for more information.

27.2 VBALOAD command

Opens the **Open** dialog box.

Shape Lite Pro Mechanical BIM

27.2.1 Description

Opens the **Open** dialog box to select a DVB or VBI file to load. After the project is loaded, use the VBARUN command to run macros within the project.

27.3 -VBALOAD command

Loads VBA projects.

Shape Lite Pro Mechanical BIM

27.3.1 Description

Loads a VBA project; works in the Command line (short for “Visual Basic for Applications”).

27.4 VBAMAN command

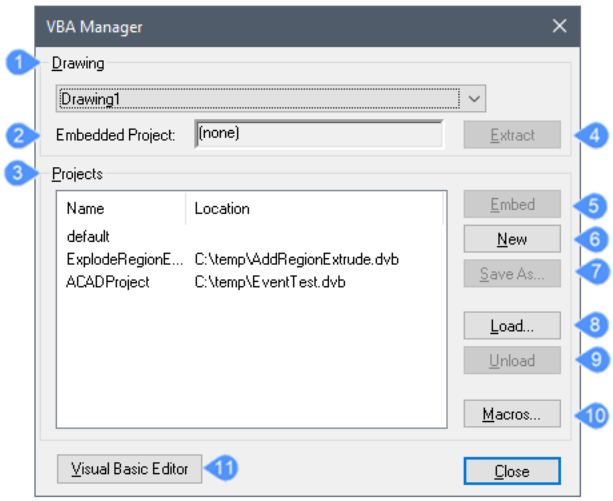
Opens the **VBA manager** dialog box.

Shape Lite Pro Mechanical BIM

27.4.1 Description

Opens the **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

27.4.2 Drawing

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.

27.4.3 Embedded Project

Displays the name of the embedded project, if any.

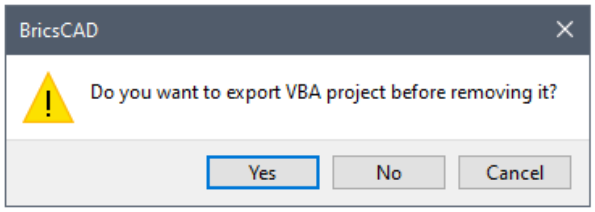
27.4.4 Projects

Name
Displays the names of projects loaded into BricsCAD at this time.

Location
Displays the drive and folder names of DVB and VBI files.

27.4.5 Extract

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.

27.4.6 Embed

Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the *Save As* dialog box.

Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the Extract option to remove the current project.

27.4.7 New

Creates a new VBA project; gives the generic name VbaProject . Use the Save As option to rename and save the DVB or VBI project file.

27.4.8 Save As

Saves the selected project by another name. Displays the *Save As* dialog box.

27.4.9 Load

Loads DVB and VBI project files; displays the *Open* dialog box.

27.4.10 Unload

Unloads the selected project without a warning.

27.4.11 Macros

Displays the *Run BricsCAD VBA Macro* dialog box (see **Run BricsCAD VBA Macro dialog box** related article).

27.4.12 Visual Basic Editor

Opens the VBA integrated development environment.

27.5 VBANEW command

Begins a new VBA project.

Shape Lite Pro Mechanical BIM

27.5.1 Description

Starts a new VBA project (short for “Visual Basic for Applications”).

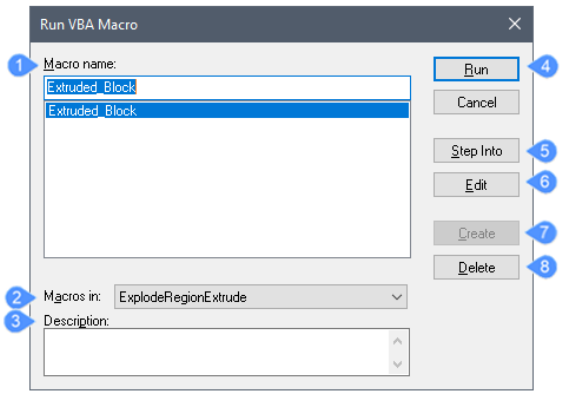
27.6 VBARUN command

Opens the **Run VBA macro** dialog box.

Shape Lite Pro Mechanical BIM

27.6.1 Description

Opens the **Run VBA macro** dialog box to run, create, edit, and delete VBA macros.



1 Macro name

2 Macros in

3 Description

4 Run

5 Step Into

6 Edit

7 Create

8 Delete

27.6.2 Macro name

Lists the names of macros found in the project file. Choose one to run.

27.6.3 Macros in

Lists the names of projects loaded into the current drawing. Choose one from the drop-down list.

27.6.4 Description

Describes the macro. You can edit this text and save it pressing the **Close** button.

27.6.5 Run

Runs the selected macro.

27.6.6 Step Into

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.

27.6.7 Edit

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.

27.6.8 Create

Creates new macros; follow these steps:

1 In the **Macro Name** field, enter a name for the new macro.

2 Click **Create**.

3 Notice the integrated development environment window. Enter the VBA code.

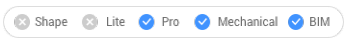
See the VBAIDE command.

27.6.9 Delete

Deletes the macro. A dialog asks to confirm the action.

27.7 -VBARUN command

Runs VBA macros.



27.7.1 Description

Runs a VBA macro defined in a DVB (drawing visual basic) file (short for ”Visual Basic for Applications”). This command is meant for use by macros.

27.8 VBASECURITY command

Opens the **Security** dialog box.

Shape Lite Pro Mechanical BIM

27.8.1 Description

Opens the **Security** dialog box to specify if VBA macros are allowed to run automatically.

27.9 VBAUNLOAD command

Unloads VBA projects.

Shape Lite Pro Mechanical BIM

27.9.1 Description

Unloads DVB (drawing visual basic) project files from the program (short for “Visual Basic for Applications”).

27.10 VERSIONCONTROL command

Enables multi-user collaboration.

Shape Lite Pro Mechanical BIM

27.10.1 Description

Enables multi-user collaboration by storing and managing projects in the cloud. Drawings are stored in the cloud using Bricsys 24/7 and checked in and out from each user's local machine.

Note: This is Beta functionality and you must request access by submitting a support request following this procedure:

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.

27.10.2 Method

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.

27.10.3 Options within the command

Init
Creates a new multi-user project in Bricsys 24/7. If you are not already logged into your Bricsys account, the **BricsCAD Analytics** dialog box displays.

Project name
Once logged in, specify the name for the new project. If the project name already exists, you are prompted to specify a different project name.

Note: Keep in mind that project names are case-sensitive.

CReate local
Creates a new local version control project by entering the path of the considered folder to create a project from.

Version control folder
Allows you to specify the folder on your local machine to set under version control. 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.

check Out
Checks out a drawing from your Bricsys 24/7 projects.

CHeck In
Checks in drawings from your local version control folder to the associated Bricsys 24/7 project.

All
Checks in all files in the version control folder.

Commit
Commits a list of modified and untracked files.

Once the files to check in are selected, a check in message can be defined.

Check in Message
Enter a relevant check in message.

Update
Syncs project drawings from your local folder with the associated Bricsys 24/7 project.

Revert
Reverts to previous state.

History
Enables you to go to older versions.

REname
Renames a drawing that is under version control.

History
Enables you to go to older versions.

27.11 VIEW command

Opens the **Drawing explorer** dialog box with **Views** selected.

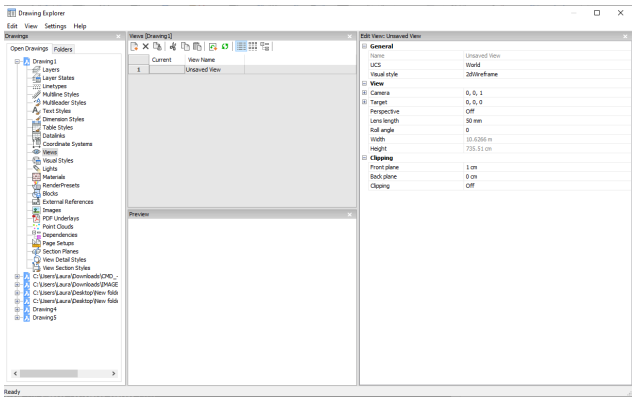
Shape Lite Pro Mechanical BIM

Icon:

Alias: DDVIEW, EXPVIEWS, V

27.11.1 Description

Opens the **Drawing explorer** dialog box with **Views** section selected to view and modify views in the current drawing.



27.11.2 Options within the Edit View panel

General

Name
Sets the name of the current view.

UCS
Assigns a UCS. When the drawing contains no saved UCS's, the droplist shows <None>. See the UCS and EXPUCS commands.

Background
Sets the background for the view. See the BACKGROUND command.

Clip Display
Assigns a clip display.

Layer Snapshot
Toggles whether to assigned the current on and off settings of layers to the view.

- Yes: assigns the current visual state of layers, so that the next time you make this view current, the layers turn themselves on or off.
- No: does not assigns the current visual state of layers, so that the next time you make this view current, the layers do not turn themselves on or off.

Visual Style
Sets the visual style. See the VISUALSTYLES command.

View

Camera
Specifies the camera point of the view in x,y,z coordinates in visual perspective mode.

Target
Specifies the target point of the view in x,y,z coordinates in visual perspective mode.

Perspective
Toggles perspective view:

- On: view is displayed in visual perspective mode
- Off: view displays in parallel perspective mode

A camera glyph is displayed in the drawing for perspective views (see the CAMERA command).

Lens Length
Sets the lens length of the camera.

Roll Angle
Rotates the camera around the view axis.

Width
Specifies the width of the view in current units.

Height
Specifies the height of the view in current units.

Clipping

Front Plane
Sets the distance between the target point and the front clipping plane.

Back Plane
Sets the distance between the target point and the back clipping plane.

Clipping
Toggles the clipping planes, which remove from any entities that are outside the planes.

27.11.3 Context Menu Options

New
Creates new named views.

Delete
Erases the named view from the drawing without warning.

Rename
Renames the view.

Select All
Selects all view definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected view as current.

27.12 -VIEW command

Controls named views.

Shape Lite Pro Mechanical BIM

Alias: -V

27.12.1 Description

Creates and sets named views in the current viewport, and deletes named views from the drawing.

Every viewport can show a different view in model and paper space.

27.12.2 Options within the command

? to list
Lists the existing views in the current drawing.

Delete
Deletes a specified view.

Orthographic
Sets a standard orthographic view point in the current viewport -- either the Top, Bottom, Front, Back, Left, or Right view.

Restore
Restores a specified view.

Save
Allows you to save the view.

Window
Save a windowed area as a named view.

27.13 VIEWBASE command

Generates associative orthographic and isometric views of 3D models in paper space layouts.

Shape Lite Pro Mechanical BIM

Icon: 

Note:

- This command is valid in Model Space only.
- Use the Tab key to select obscured entities.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for generated views are updated automatically when the 3D model is modified.
- Setting the GENERATEASSOCATTRS (Generate associative attributes) system variables ON allows VIEWBASE to generate drawings in which dimensions and tags are updated automatically when the 3D model has been modified.
- The DRAWINGVIEWQUALITY system variable defines the quality of drawing views.

- When the DRAWINGVIEWFLAGS system variable is set to 1, the parallel generation and update of views is enabled. This can reduce view processing time, but uses more resources.
- When the DRAWINGVIEWFLAGS system variable is set to 2, user interaction during the generation of drawing views is allowed.
- The DRAWINGVIEWASM system variable enables the use of assembly data structures to optimize generation of drawing views.
- When the DRAWINGVIEWENTS system variable is ON, points are displayed in generated drawing views in accordance with PDMODE and PDSIZE system variables settings.

Tip: This command can be entered transparently during commands ('viewbase).

27.13.1 Method

Select one or more entities (3D solids, blocks, components) or press Enter to select all 3D entities in the Model Space from which to generate the drawing views in a layout tab or choose an option. Hit the Tab key to select nested entities.

Enter the name of a new or existing layout or press Enter to accept the current layout.

The command switches to the layout tab where a point must be picked to define a position for the base view or enter an option.

Select the position for each projected view by moving the cursor. Depending on the position of the cursor with respect to the base view one of five orthogonal views (top, left, right, back and bottom) and four isometric views can be inserted. The views are aligned automatically depending on the selected projection type (see above).

Tap the Ctrl key to toggle alignment On and Off. When it is Off, you can place the current view in any location.

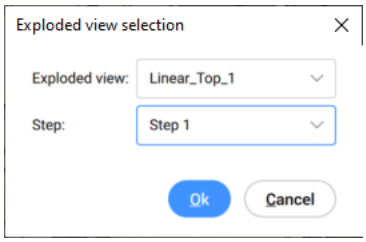
27.13.2 Options within the command

Entire model
Selects all the 3D entities in model space.

preseTs
Specifies the types of generated drawings and their placement in the layout; displays the **Drawing View Presets** dialog box.
The selected preset is saved through the DRAWINGVIEWPRESET system variable.

Special views
Selects a style of exploded representations, if any exist in the drawing.

Exploded view
Creates an exploded drawing view if an exploded representations exists in the drawing.
The created views correspond to the selected step when selected. For "<All steps>", the views should correspond to the final state of the exploded view (last step).



Exploded view
From the drop-down list, choose a style for the exploded representation.

Step
From the drop-down list, choose a step for the exploded representation.

- For "<All steps>", the views correspond to the final state of the exploded view (last step).
- For a specified step, the views correspond to the selected step.

Unfolded view
Creates an unfolded drawing view for solids with associated unfolded view.

Back
Returns to the previous prompt.

Scale
Sets the Scale property of the paper space viewports for the various views:

fit 4 views
The scale is adjusted to fit the four standard orthographic views: Front, Top, Left, Right. The Front view (base view) is defined by the Orientation option.

fit 9 views
Adjusts the scale to fit five orthographic views and four isometric views.

fit 5 views
Adjusts the scale to fit five orthographic views: Front, Top, Left, Right, Back.

fit 10 views
Adjusts the scale to fit six orthographic views and four isometric views.

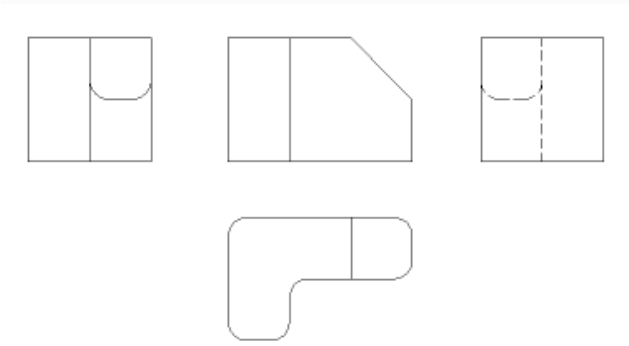
Standard scales
Displays the scale list as maintained by the SCALELISTEDIT command; select a scale from the list.

Custom scales
Prompts you to type a scale in the Command line.

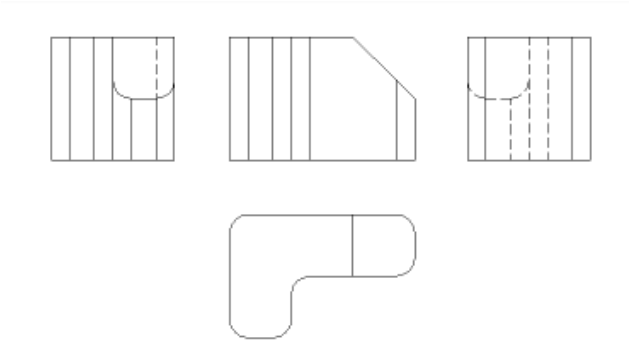
Hidden Lines
Controls the visibility of hidden lines.
When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent Lines
Toggles whether tangent edges between tangent faces are created. Visible tangent edges are created on the BM_Tangent_Visible layer. Hidden tangent edges are created on the BM_Tangent_Hidden layer.

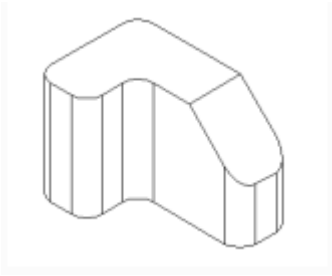
No
Does not display tangent lines.



Yes
Displays the tangent lines.



Note: Tangent edges are always created for isometric views. Freeze or switch off the display of the BM_Tangent_Visible layer to hide tangent edges in isometric views.



INterferance edges
Toggles between ON and OFF the visibility of interference edges between solids which intersect with each other. When it is ON, a line is drawn where the solids are met.

trailiNg lines
Controls the visibility of trailing lines.
To create exploded views, use the BMEXPLODE command; the model must have inserts of mechanical components to use this command.

Orientation
Defines the base view orientation. Rotates the 3D model so that the main view is projected on the vertical projection plane (V.P.).

Projection type
Defines the layout of the views.

First angle
It is also known as European projection.

Third angle
It is also known as American projection.
The default projection type depends on the value of the MEASUREMENT system variable.

Isometric Geometry
Defines the style for the isometric views: rendered 3D view or 2D drawing:

2D views
Draws isometric views as 2D drawings.

3D views
Draws isometric views as 3D solids with the Conceptual visual style applied.

sElect
Selects additional entities to include or exclude.

Remove
Removes entities from drawing views.

Entire model
Includes all entities from model space in the drawing views.

Layout
Opens the previous layout with updated drawing views.

27.14 VIEWBREAK command

Creates a broken view on drawing views generated by the VIEWBASE command in a paper space layout.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command operates only in Paper Space.

Note: This command can be entered transparently during commands (‘viewbreak)

27.14.1 Method

Select the drawing view from which to generate the broken view by clicking inside the drawing view. Select the first and the second points which specifies the first and the second planes of the cuts.

The default direction of the symbol is based on the size of the viewport:

- Vertical if the viewport is wider than high.
- Horizontal if the viewport is higher than wide.

27.14.2 Options within the command

Type
Allows to set the broken symbol type:

sTraight
Line geometry. Supports the Gap distance property.



Spline
Spline geometry. Supports the Gap distance, Width and Height.



Zigzag
Spline geometry. Supports the Gap distance, Width and Height properties.

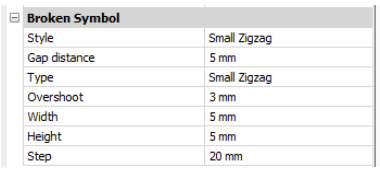


sMall zigzag
Supports the Gap distance, Width, Height and Step properties.



Current
Use the last used broken symbol type.

Note: The properties of the Broken Symbol can be edited in the Properties panel:



Broken Symbol	
Style	Small Zigzag
Gap distance	5 mm
Type	Small Zigzag
Overshoot	3 mm
Width	5 mm
Height	5 mm
Step	20 mm

Style/Type
Select the style / type in the drop-down list.

Gap distance
Defines the distance* between the two parts of the broken symbol.

Overshoot
Defines the length of the extends for the break lines outside of the 2D view.

Width
Defines the width* of the shape symbol in the symbol direction.

Height
Defines the height* of the shape symbol in the direction orthogonal to the symbol direction.

Step
Defines the distance* between the small zigzag symbols.

* Expressed in layout units.

Vertical
Aligns the symbol along the Y axis.



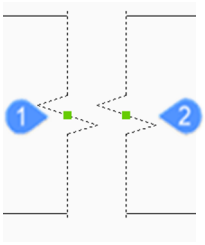
Horizontal
Aligns the symbol along the X axis.



27.14.3 Edit Grips

You can edit break symbol grips.

Select the break symbol and 2 Grips display:



1 Controls the position of the first break plane in the model space

2 Controls the position of the second break plane in the model space

27.15 VIEWDETAIL command

Draws view details of drawing views made with the VIEWBASE command.

Shape Lite Pro Mechanical BIM

Icon:

Note:

- This command operates only in Paper Space.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for detailed views are updated automatically when the 3D model is modified.
- DRAWINGVIEWQUALITY variable defines the quality of drawing views.
- The AUTOVPFITTING variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.

27.15.1 Method

Select the drawing view from which to extract the detail by clicking inside a drawing view. The program highlights the selected view.

After that, specify the center point of the detail view by picking a point inside the parent view and the style of the view boundary.

There are two boundary types from which you can choose:

- Circular - draws a circular boundary.
- Rectangular - draws a rectangular boundary.

Note: You may always switch the boundary type between circular and rectangular using the Boundary option.

Specify the position for detail view by picking a point in the layout, away from the parent view.

27.15.2 Options within the command

Scale
Specifies the scale of the detail view, which by default is twice the parent viewport scale:

Standard scales
Choose a standard scale in the list; the list can be edited by the SCALELISTEDIT command.

Custom
Specify a custom scale factor.

Relative custom scale
Calculates the scale factor relative to parent view by multiplying the scale factor of the source view with this number.

from Parent
Sets the scale of the section view equal to the scale of the parent view.

Hidden lines
Control the visibility of hidden lines, or uses the same hidden lines setting as the parent view.

Note: When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent lines
Controls the display of tangent edges which appear in the transition from a flat face and a curved face, such as with fillets:

Note: When the BM_Tangent_Visible layer is off or frozen, tangent lines are not display.

anChor
Determines if the center of the viewport is anchored so that the viewport grows and shrinks around its center point or not.

Geometry
Selects the visual style for the section view:

2D
Section view uses the 2dWireframe visual style.

3D
Section view uses a rendered visual style. This is Conceptual by default. Use the Properties panel to choose a different visual style.

Annotation
Determines the annotations to use.

Identifier
Specifies the view detail identifier by entering a name for the view detail.

Label
Toggles the display of the view detail label.

Boundary
Toggles the style the boundary between circular and rectangular.

Circular
Changes the boundary to a circle.

Rectangular
Changes the boundary to a rectangle.

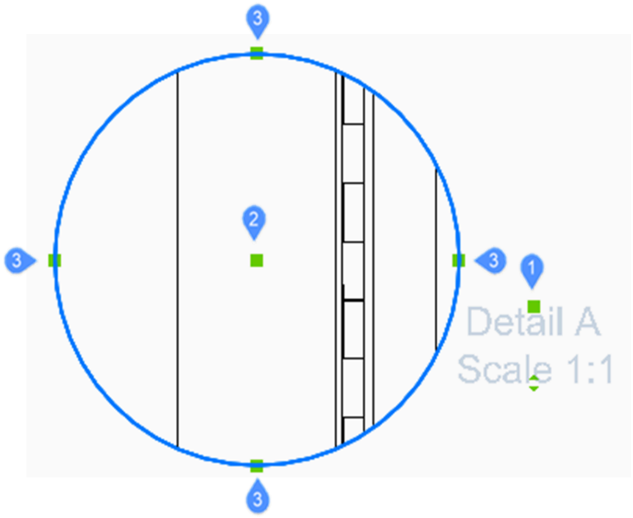
modeL edge
Controls whether a connection line is drawn between the detail view and the detail boundary in the parent view.

smooth with Border
Does not draw a line.

smooth with Connection line
Draws a connecting line from the detail view to the parent view.

27.15.3 Grips Editing

Detail views can be edited through grips: select the view and six grips display:



1 Controls the position of the identifier.

2 Controls the position of the section detail.

3 4 grips control the size of the detail boundary.

27.16 VIEWDETAILSTYLE command

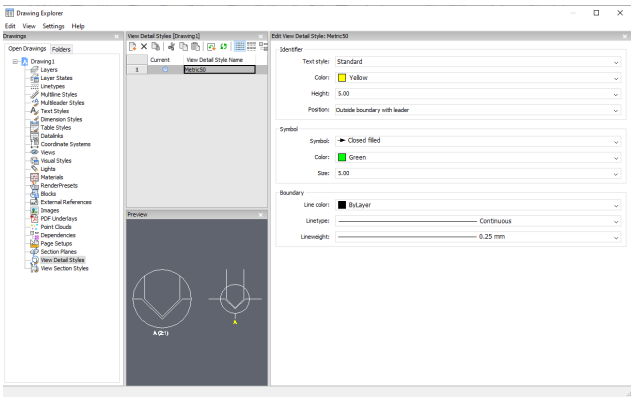
Opens the **Drawing explorer** dialog box with **View Detail Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.16.1 Description

Displays the **Drawing Explorer** dialog box with the **View Detail Style** category selected to create and modify view detail styles.



27.16.2 Options within the Edit View Detail Style panel

Identifier
Defines the properties of the view detail identifier.

Text style
Specifies the text style used by the identifier text.

Color
Specifies the color of the identifier.

Height
Specifies the height of the identifier.

Position
Determines where the identifier is located.

Outside boundary
Places the identifier outside the detail's boundary. No symbol is used.

Outside boundary with leader
Places the identifier outside the detail's boundary and draws a leader from the parent view to the detail. No symbol is used

On boundary
Places the identifier on the detail's boundary.

On boundary with leader
Places the identifier on the detail's boundary and draws a leader from the parent view to the detail.

Symbol
Defines the properties of the symbol.

Symbol
Specifies the look of the symbol.

Color
Specifies the color of the symbol.

Size
Specifies the size of the symbol.

Boundary
Defines the properties of the detail viewport boundary.

Line color
Specifies the color of the boundary line

Linetype
Specifies the line type of the boundary.

Lineweight
Specifies the weight of the boundary line.

27.16.3 Context Menu Options

New
Creates new named view detail styles.

Delete
Erases the named view detail style from the drawing without warning.

Rename
Renames the view detail style.

Select All
Selects all view detail styles definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected view detail style as current.

27.17 VIEWEDIT command

Changes the scale and hidden line visibility of a drawing view(s) if the view(s) was created with the VIEWBASE command.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command operates only in Paper Space.

27.17.1 Method

Select the drawing view(s) for modifying by clicking inside one or more drafting viewports or press Enter to select all drawing views in the current layout.

The next prompt will vary depending on the selected view type. When different view types are selected, only the shared options are available.

27.17.2 Options within the command

Standard view options: see the VIEWBASE command

Section view options: see the VIEWSECTION command

Detail view options: see the VIEWDETAIL command

27.18 VIEWEXPORT command

Exports drawing views.

Shape Lite Pro Mechanical BIM

Icon: 

27.18.1 Description

Exports drawing views generated by the VIEWBASE , VIEWSECTION or VIEWDETAIL commands to the model space of the current drawing, via the Clipboard to a new drawing or as a DWG/DXF file on disc.

27.18.2 Method

Click inside one or more viewports or select All views and choose the destination of the views (see options).

27.18.3 Options within the command

Clipboard

Exports the selected view(s) to the clipboard, which can then be pasted into any drawing.

File

Exports the selected views to a DWG or DXF file; displays the Save Exported Views dialog box.

Model space

Exports the view to the current drawing's model space.

Note: Specify if remove exported views:

- Yes - the viewports of exported views are removed from the layout.
- No - viewports of exported views are kept in the layout.

27.19 VIEWHORIZONTAL command

Rotates viewpoint to the horizontal.

Shape Lite Pro Mechanical BIM

27.19.1 Description

Sets the Z component of the 3D viewpoint to 0 in the current UCS, so that the viewpoint rotates to the horizontal.

This command works like setting Z to 0 with the VIEWPOINT command.

27.20 VIEWLABEL command

Shape Lite Pro Mechanical BIM

27.20.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

27.21 VIEWPOINT command

Changes the 3D viewpoint.

27.21.1 Description

This is an alias for VPOINT command.

27.22 VIEWPROJ command

Generates additional projected views from an existing drawing view previously created by the VIEWBASE command (short for "view projection").

Shape Lite Pro Mechanical BIM

Icon: 

Note:

- This command operates only in Paper Space.
- The DRAWINGVIEWFLAGS system variable allows to enable parallel creation or updating of drawing views. This can reduce view processing time, but uses more resources.

27.22.1 Method

Select the view from which to generate new projected views and select the position of the new projected views.

Depending on where you move the cursor, you can place up to five orthogonal and four isometric views. The views are aligned automatically. Press Ctrl key to toggle alignment on and off to place the view anywhere on the sheet.

27.22.2 Options within the command

Isometric geometry

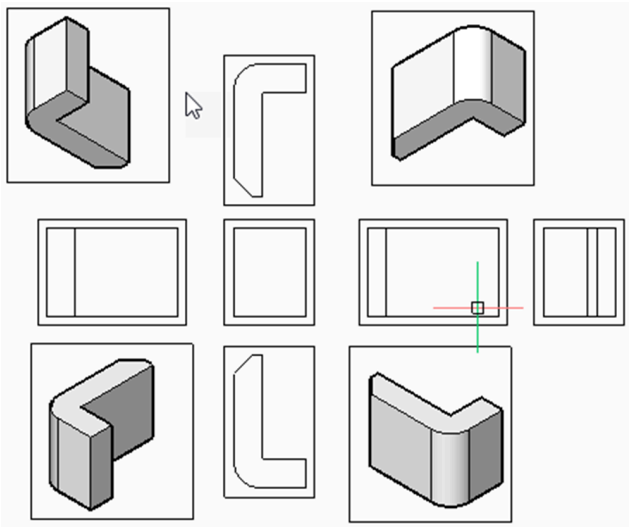
Sets the style for isometric views.

2D views

Draws isometric views as 2D projections of the 3D geometry and turns on the BM_Hidden layer so that hidden lines are shown as dashed lines.

3D views

Draws isometric views as 3D geometry and sets the visual style of the viewport.

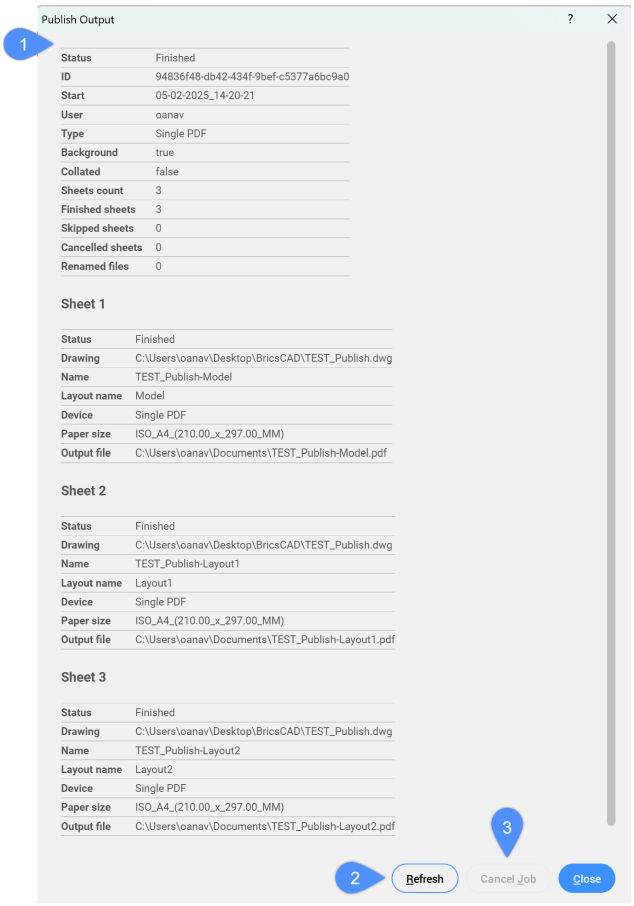


27.23 VIEWPLOTDETAILS command

Displays details about publish jobs.

27.23.1 Method

Opens the **Publish Output** dialog box that allows you to view the details about publish jobs, the errors that have occurred, or to cancel the ongoing publish jobs.



Publish Output Details (1)

Displays detailed information about completed publish jobs or the errors that have occurred. You can also monitor the ongoing publish jobs.

Note: Check the count to ensure that all sheets were properly processed. The following counts are displayed:

- **Sheets count**: displays the total number of sheets specified for printing.
- **Finished sheets**: displays the number of sheets that were printed.
- **Skipped sheets**: displays the number of sheets that could not be printed.
- **Cancelled sheets**: displays the number of sheets that were cancelled.
- **Renamed files**: displays the number of renamed files. This means the sheet was printed but the name was modified to avoid name collision or disallowed characters.

Refresh (2)

Refreshes the list of published jobs to see the progress before the process is finished.

Cancel Job (3)

Stops the ongoing publish jobs.

Note: It is only available when the publish process is in progress. When the process is completed or cancelled, the button becomes greyed out.

27.24 VIEWRES command

Sets viewing resolution.

Shape Lite Pro Mechanical BIM

27.24.1 Description

Sets the view resolution for curved entities; also toggles fast-zoom mode (short for "view resolution").

27.24.2 Options within the command

Choose whether to use redraws or regenerations for view changes, such as zooms.

Choose the level of quality for displaying 2D curves, such as circles and arcs (range 1-20000)

- 1 - curves like circles and arcs may look like polygons, but display at a fast speed.
- 100 - default value.
- 20000 - curves almost always look round, but at a slower display speed.

27.25 VIEWSECTION command

Creates cross-section views of drawing views generated with the VIEWBASE command in a paper space layout.

Shape Lite Pro Mechanical BIM

Icons: 

Note:

- This command operates only in Paper Space.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for section views are updated automatically when the 3D model is modified.
- If the SECTIONABLE property of a mechanical component is OFF, the component appears non-sectioned in the section views of the Full section type.
- DRAWINGVIEWQUALITY variable defines the quality of drawing views.
- The AUTOVPFITTING variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
- This command can be entered transparently during commands (‘viewsection)

27.25.1 Method

Select the drawing view to generate the section by clicking inside a drawing view. The program highlights the selected view. Create a section and choose a location for the result view.

27.25.2 Options within the command

Select type

Control the shape of the section plane:

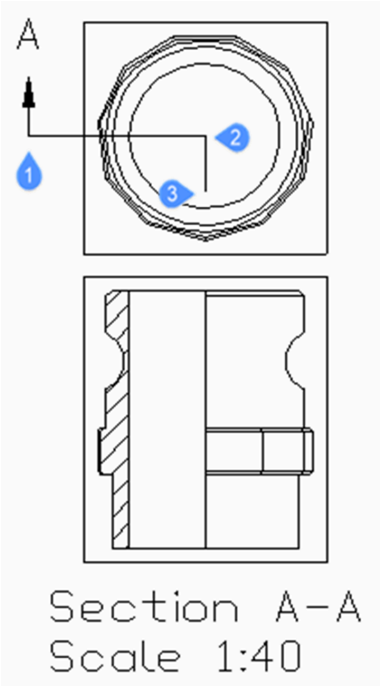
Full

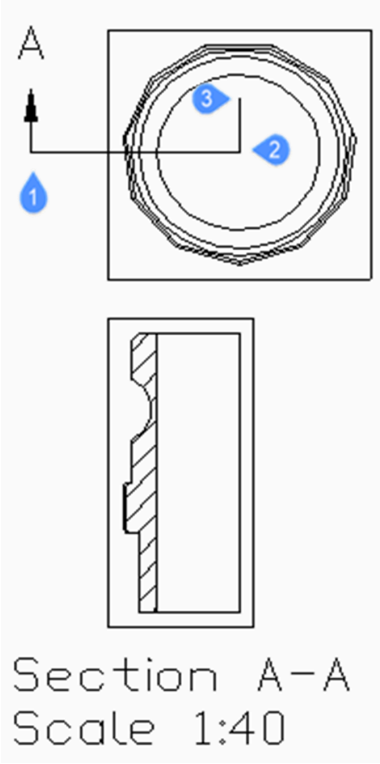
The section line defines an infinite plane that cuts through the entire model.

Half

The section line defines a half-plane that cuts part of the model. You must specify by clicking the start point (1) of section line, the second point (2) to define the limit of the half plane and the third point (3) to define the view direction.

The position of the third point determines whether the part of the model that is not cut displays (left) or not (right).





Offset
The section line defines a series of cutting regions located on different offsets from each other. Enter Done to complete the section line.

Aligned
The section line defines a polyline, where every segment defines a cutting region. The resulting section will have the length equal to the sum of the lengths of cutting regions. Enter Done to complete the section line.

Scale
Specifies the scale of the detail view, which by default is twice the parent viewport scale.

Standard scales
Choose a standard scale from the list; the list can be edited by the SCALELISTEDIT command.

Custom
Specify a custom scale factor.

from Parent
Sets the scale of the section view equal to the scale of the parent view.

Hidden lines
Control the visibility of hidden lines or uses the same hidden lines setting as the parent view.

Note: When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent lines
Controls the display of tangent edges which appear in the transition from a flat face and a curved face, such as with fillets.

Note: When the BM_Tangent_Visible layer is off or frozen, tangent lines are not display.

anChor
Determines if the center of the viewport is anchored so that the viewport grows and shrinks around its center point or if the geometry is fixed.

Geometry
Selects the visual style for the section view.

2D
The section view uses the 2dWireframe visual style.

3D
The section view uses a rendered visual style. This is Conceptual by default. Use the Properties panel to choose a different visual style.

Annotation
Determines the annotations to use.

Identifier
Specifies the view detail identifier by entering a name for the view detail.

Label
Toggles the display of the view detail label.

Depth
Specifies the view depth of a section view.

Full
Sets the depth to the extents of the model (maximum view depth).

Custom
Limits the view depth by entering the depth distance (type a positive value or move the cursor to define the view depth dynamically).

Projection
Determines how the section is projected.

Normal
Draws the section as a union of projections from every cutting region (each segment of the section polyline) in its normal direction.

Orthogonal
Draws the section in the normal direction of the first section region (the first segment of the section polyline).

Rotate view
Allows you to rotate the drawing view viewport but does not rotate the frame.

Horizontal
Rotates a segment horizontally.

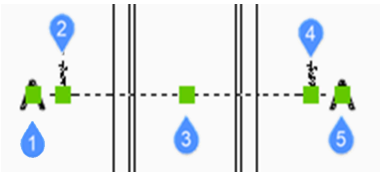
Vertical
Rotates a segment vertically.

Custom angle
Allows you to specify the angle of alignment.

27.25.3 Grips Editing

You can edit section lines with grips.

Selecting either the section line, one of the identifiers or an arrow, 5 Grips display:



1 Controls the position of the first identifier.

2 Defines the start point of the section line.

3 Allows you to move the section line.

4 Defines the endpoint of the section line.

5 Controls the position of the second identifier.

27.26 VIEWSECTIONSTYLE command

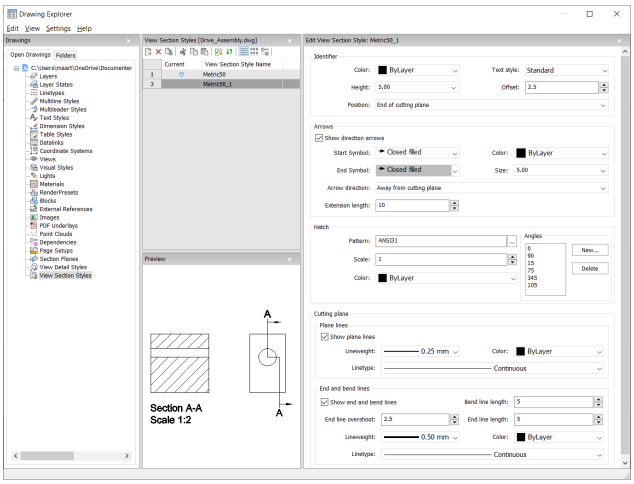
Opens the **Drawing explorer** dialog box with **View Section Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.26.1 Description

Opens the **Drawing Explorer** dialog box with **View Section Styles** category selected to view and modify view section styles in the current drawing.



27.26.2 Options within the View Section Style panel

Identifier
Sets the properties of the section identifier.

Color
Specifies the color of the identifier.

Text Style
Specifies the text style used by the identifier text. To employ a different text style, use the STYLE command to create it.

Height
Specifies the height of the identifier.

Offset
Specifies the distance from the end of the cut lines to the arrow.

Position
Determines where the identifiers are located.

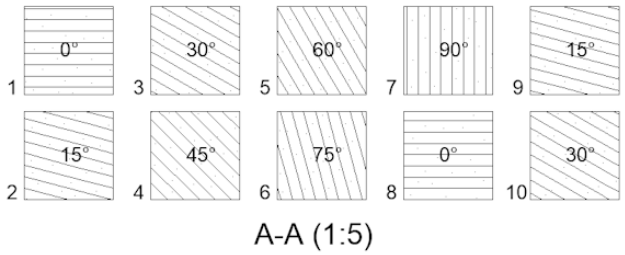
- **End of cutting plane**: places the identifiers at the ends of the cutting planes.
- **Above direction line**: places the identifiers outside the detail's boundary and draws a leader from the parent view to the detail. No symbol is used.
- **Above direction symbol**: places the identifiers above the direction symbols.
- **Start of direction arrow**: places the identifiers at the starts of the arrows.
- **End of direction arrow**: places the identifiers at the ends of the arrows.

Arrows
Sets the properties of the section arrows.

- Show direction arrows: toggles the display of the arrows
- Start Symbol: specifies the look of the start symbol
- Color: specifies the color of the arrow
- End Symbol: specifies the look of the end symbol
- Size: specifies the size of the arrow
- Arrow direction: points the arrow towards or away from the cutting line
- Extension length: determines the length of the "dimension" line on the arrows

Hatch
Sets the properties of the section hatching.

- Pattern: choose a pattern name from the drop-down list.
- Browse: displays the Hatch Pattern Palette dialog box, from which you choose a pattern visually.
- Angles: sets the angle for the pattern, one angle for subsequent cuts. The first angle in the list is applied to the first solid, the second angle to the second, and so on. When there are more solids to cut than angles listed, then the program starts again from the first angle. See figure below.



- New: adds angles to the list. Displays the New Hatch Angle dialog box, in which you enter an angle.
- Delete: deletes the selected angle from the list.
- Scale: specifies the scale of the pattern.
- Color: specifies the color of the pattern.

Cutting Plane | Plane Lines
Sets the properties of the cutting plane lines.

- Show plane lines: toggles the display of the lines.
- Lineweight: specifies the weight of the lines.
- Color: specifies the color of the lines.
- Linetype: specifies the pattern of the lines.

Cutting Plane | End and Bend Lines
Sets the properties of the end and bend lines.

- Show end and bend lines: toggles the display of the lines.
- Bend line length: specifies the distance the bend line offsets from one cut line to another.
- End line overshoot: specifies the distance from the section to the end of the plane line.
- End line length: specifies the length of the end lines.
- Lineweight: specifies the weight of the lines.
- Color: specifies the color of the lines.
- Linetype: specifies the pattern of the lines.

27.26.3 Context Menu Options

New
Creates a new view detail style as a copy of the currently selected style.

Delete
Removes the selected style from the drawing.

Rename
Renames the selected view section style.

Select All
Selects all view section style definitions.

Invert selection
Deselects the current selection and vice versa.

27.27 VIEWUPDATE command

Updates drawing views.

Shape Lite Pro Mechanical BIM

Icon: 

Note:

- The DRAWINGVIEWFLAGS system variable enables parallel creation or updating of drawing views. This can reduce view processing time, but uses more resources.
- The DRAWINGVIEWASM system variable enables the use of assembly data structures to optimize generation of drawing views.

27.27.1 Description

Manually updates the selected or all drawing views created by the VIEWBASE and VIEWSECTION commands when automatic updates (VIEWUPDATE) are turned off.

27.27.2 Options within the command

Select drawing views
Updates the selected views.

All views
Updates all views in the current layout.

27.28 VISIBILITYSTATES command

Creates and edits visibility states in the Command line.

Shape Lite Pro Mechanical BIM

Icon: 

27.28.1 Description

Creates and edits visibility states via Command line. It also opens the **Visibility States** panel if it was closed before running the VISIBILITYSTATES command.

Note: BricsCAD visibility states are not compatible with AutoCAD® visibility states. When you open a drawing with BricsCAD parametric blocks in AutoCAD®, they are displayed as they were saved in BricsCAD. However, when such parametric blocks are edited or copied to another drawing in AutoCAD®, all entities in the block become visible.

27.28.2 Options within the command

New parameter
Creates a new parameter.
Type a name for the parameter (<P>), then press Enter.

Enter new state for <P>
Type a name for the new parameter state (<S>).

Add entities
Adds entities to the specified state of a parameter.

Select entities for <P>=<S> [selection options (?)]
Select entities for the new state using any selection method.
Press Enter to stop selecting.

When the parameters and states already exist in the drawing, the existing parameters and states are added to the prompts:

New parameter or [<P> (1)/ [<P> (2) …]
Do one of the following:

- Type a name for a new parameter.
- Type the number of the parameter you want to edit.

Enter new state for [<P> or [<S> (1)/ [<S> (2) …]
Do one of the following:

- Type a name for a new state of the parameter.
- Type the number of the state you want to edit.

Remove entities
Removes entities from the specified state of a parameter.

make entities Visible
Makes entities visible to the specified state of a parameter.

make entities Invisible
Makes entities invisible to the specified state of a parameter.

Select state
Makes the mentioned state active.

Note:

- <P> is a placeholder for the name of the visibility parameter entered in the previous step.
- <S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, the actual names of the parameter and the state will be displayed instead of <P> and <S>.

27.29 VISIBILITYSTATESPANELCLOSE command

Closes the **Visibility States** panel.

Shape Lite Pro Mechanical BIM

Icon: 

27.29.1 Methods

The VISIBILITYSTATESPANELCLOSE command closes the **Visibility States** panel.

For more information on viewing and managing the visibility parameters and states see the article **Visibility States** panel.

27.30 VISIBILITYSTATESPANELOPEN command

Opens the **Visibility states** panel.

Shape Lite Pro Mechanical BIM

Icon:

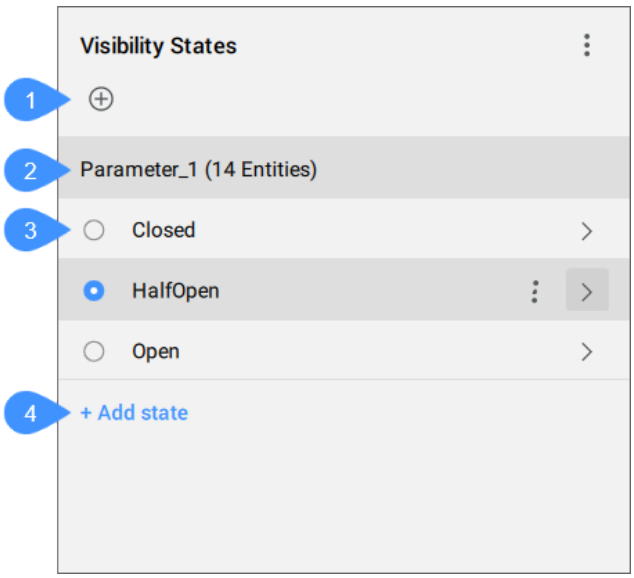
27.30.1 Description

Opens the **Visibility States** panel to display it in the current workspace. The **Visibility States** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Visibility States** panel can be either floating, docked or stacked.

Visibility states parameters allow you to create blocks where just selecting one or more parameters change the appearance of the block by showing or hiding various entities inside the block. Each visibility states parameter can have one or more states.

Note: There is no need to be in the **Block editor** to use Visibility States.

The **Visibility States** panel displays the visibility parameters found in the drawing and their corresponding visibility states.



1 Add parameter

2 Parameter's name

3 List of parameter's visibility states

4 Add state

Add parameter

Adds a new parameter to the block.

Note: The new created parameters appear in the **Parameters** section in the **Properties** panel.

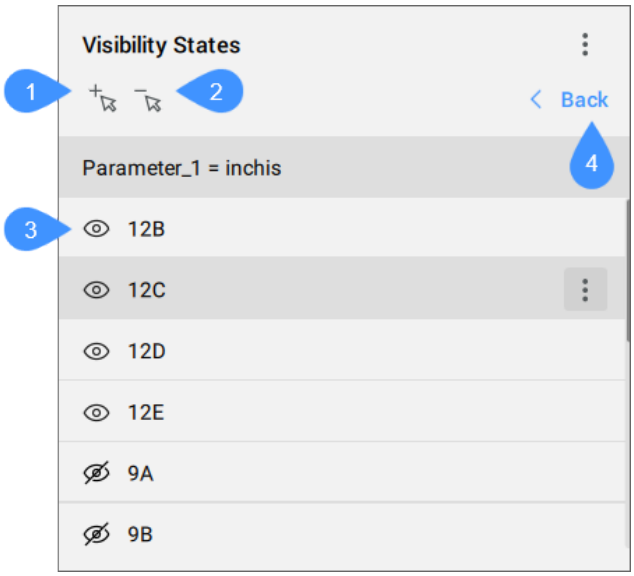
Parameter's name
Double-click to edit the parameter's name.

List of parameter visibility states
Lists all the parameter's visibility states. Double-click a state name or click the right-side arrow to access its entities.

Add state
Adds a new visibility state to the parameter. If the parameter already has states with entities assigned, all these entities will, by default, be invisible for the new state.

27.30.2 State entities

After selecting a visibility state, the panel shows the list of entities assigned to that state.



1 Add entities

2 Remove entities

3 List of entities

4 Back button

Add entities
Allows you to select entities in the drawing and add them to this parameter.
Entities that are not in the parameter are added and made visible only for the selected state, and invisible for all other states. Entities that are already in the parameter are made not affected.

Remove entities
Allows you to select entities in the drawing and remove them from this parameter and all its states.

Note: If entities are already selected before adding or removing them, the actions apply on the already selected entities.

List of entities
Lists all the entities that belong to the selected visibility state, along with an icon that indicates the visibility of each of these entities.

Note: One entity can be assigned to multiple parameters at the same time. In this case, the entities will only be visible if they are set to be visible for the current states of both parameters.

If you select one or more entities, then click one entity's 'eye' icon, its visibility is switched (from visible to invisible or vice versa). The new visibility also applies to all other selected entities.

Note: If multiple entities were selected and the eye icon was clicked for one entity that is currently invisible, all selected entities will become visible.

Whenever there are entities that are (in)visible for all states of a parameter, a message label will appear at the bottom of the main view of the panel, together with two link buttons. **Show** will highlight these entities in red and, if they are invisible, make them visible and semi-transparent. **Remove from Parameter** will remove these entities from the appropriate parameters, making them visible but no longer linked to the parameter.
The selection of entities in the panel changes the selection in the drawing, and vice versa. This allows you to easily check which entities are visible or invisible in the current state.

Back button
Takes you back to the main view of the panel.

27.30.3 Context menu options

Show invisible
This option can be toggled on and off. When on, any entities that are currently made invisible by the selected state are highlighted in blue and are semi-transparent.

Note: This enables you to perform any actions for which you might need to select both visible and invisible entities at the same time (move, align, remove from parameter).

Note: This option stays active until you close the panel, switch to a new document or enter block edit mode.

Options for Parameter

Remove
Removes the parameter and its states. Any entities that were made invisible by this parameter will become visible again.

Rename
Allows you to type a new name to replace the existing one.

Note: As for parameters and constraints, there are limitations on which characters are allowed in a parameter.

Remove entities from parameter
Removes the selected entities from the parameter.

Options for State

Clone
Creates a copy of the state.

Remove
Removes the state from the parameter.

Note: This option does not affect the other states of the parameters.

Rename
Type a new name to replace the existing one.

Note: There are no limitations on which characters are allowed.

Make entities visible
Makes the selected entities visible.

Note: Entities that are not in the parameter are added and made visible only for the selected states. Entities that are already in the parameter are made visible for the selected state, without affecting the other states.

Make entities invisible
Makes the selected entities invisible.

Note: Entities that are not in the parameter are added and made invisible only for the selected states and visible for the others. Entities that are already in the parameter are made invisible for the selected state, without affecting the other states.

27.31 VISIBLE command

Shape Lite Pro Mechanical BIM

27.31.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

27.32 VISUALSTYLES command

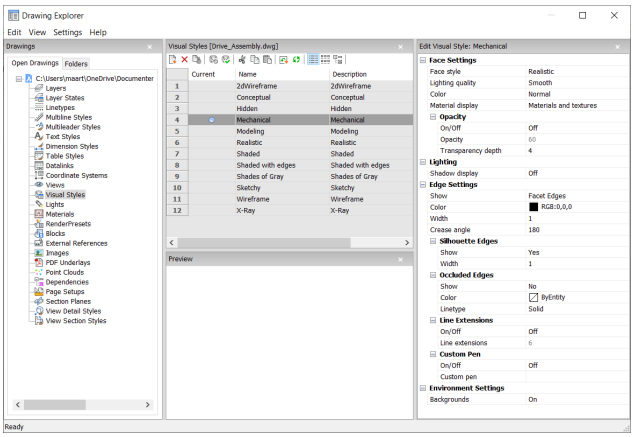
Opens the **Drawing explorer** dialog box with **Visual Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.32.1 Method

Opens the **Drawing explorer** dialog box with **Visual Styles** category selected to view and modify visual styles in the current drawing.



Reset to default ( )
Restores the original settings for predefined visual styles.

Apply visual style ( )
Applies the selected visual style to the current viewport.

Edit visual style
Displays the settings of the selected visual styles. To edit a setting: click the settings field, then type a new value or select an option in the drop down list.

Face Settings
Specifies the look of faces of 3D models.

Face style
Selects an overall color scheme for the visual style.'

Lighting quality
Specifies the quality of curved faces.

Color
Selects the color mode.

Material display
Determines whether materials are displayed. Materials are applied with the MATERIALASSIGN command. When **Material Display** is not off, then highlights and opacity are ignored, as materials provide their own values for them.

Opacity
Specifies the level of transparency of faces.

On/Off
Toggles the opacity on and off.

Opacity
Specifies the level of transparency (not available when materials are on).

Transparency depth
Specifies how far transparency reaches in terms of the number of overlapping entities.

Lighting
Determines whether shadows are displayed.

Shadow display
Specifies how shadows are displayed:

- **Off**: objects cast no shadows, the preferred option as shadow-casting slows down the program's display speed, so leave it turned off unless you are saving images to file or plotting them.
- **Ground shadows**: objects cast shadows on the shadow plane, but not on each other
- **Mapped object shadows**: objects cast shadows on the ground and on each other

Edge Settings
Specifies the style of edge to display.

Show
Specifies how edges are displayed.

- **None**: neither facets nor isolines nor edges are displayed; this setting turns off many other settings.
- **Isolines**: isolines and edges are displayed; isolines are the curved lines that simulate the surfaces of curved faces.
- **Facet Edges**: facets and edges are displayed on entities.

Color
Specifies the color for all edges.

Width
Specifies the width of edge lines.

Number of lines
Specifies the number of isolines drawn on curved surfaces, from 0 to 2047.

Always on top
Determines if all isolines are displayed, or only those that are "on top" (hidden isolines being hidden).

Silhouette Edges
Determines how fat outlines are around models.

Show
Toggles the display of silhouette edges.

Width
Specifies the width of the silhouette edges in the range of 1 to 25 pixels; applies to all entities in the view-port equally.

Occluded Edges
Specifies how to handle occluded (hidden) edges and facets; this setting allows you to display hidden lines in a different color and linetype from visible lines.

Show
Toggles the visibility of occluded edges and facets.

Color
Specifies the color for visible obscured edges and facets. Click **Select color...** to select another color from the **Color** dialog box.

Linetype
Specifies the line type for visible obscured edges and facets. The program does not use the usual linetype patterns, but a separate set. All these settings do not apply to isolines.

Line Extensions
Extends edges beyond their boundaries, also known as "overhang".

On/Off
Turn line extensions on or off.

Line extensions
Specifies the distance over which the lines extend beyond boundaries, such as face edges.

Custom Pen
Determines the use of a custom line drawing style.

On/Off
Turn the drawing style on or off.

Custom pen
Specifies the style of line drawing.

Environment Settings
Toggles the display of a background in the viewport.
Backgrounds consist of either a solid color, a gradient of two or three colors, or a raster image, and are set by the BACKGROUND command.

Note: The ANTIALIASSCREEN system variable controls the amount of anti-aliasing (edge smoothing) applied during on-screen rendered mode display. The default value is 1, maximum value is 5. High anti-alias values incur a high calculation cost.

27.32.2 Context Menu Options

New
Creates a new visual style.

Delete
Deletes the selected visual style, except for predefined visual styles such as 2dWireframe, 3D Hidden, 3dWireframe, Conceptual, Realistic…

Reset to default
Restores the selected visual style(s) to default.

Apply to current viewport
Applies the selected visual style to the current viewport.

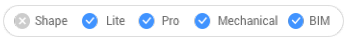
Rename
Renames the selected visual style.

Select All
Selects all visual style definitions.

Invert Selection
Deselects the current selection and vice versa.

27.33 -VISUALSTYLES command

Manages visual styles.



27.33.1 Description

The command sets and manages visual styles at the Command line.

27.33.2 Options within the command

set Current
Sets a visual style for the current viewport. The options are:

- 2dwireframe
- Wireframe

- Hidden
- Realistic
- Conceptual
- Shaded
- shaded with Edges
- shades of Grey
- SKetchy
- X-ray
- Other
- cUrrent

Note:

- **Other** option allows you to select a custom visual style. For example, BIM templates have specific predefined visual styles, Bim, Maquette, Render and See-through.
- **cUrrent** option sets the current visual style.

Save as
Saves the current visual style by a new name when you make changes to the properties of a visual style.

Note: **2dWireframe** visual style cannot be saved.

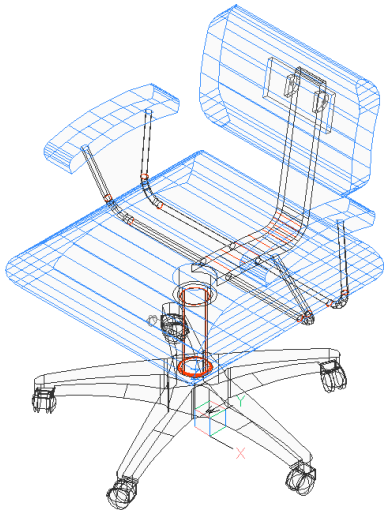
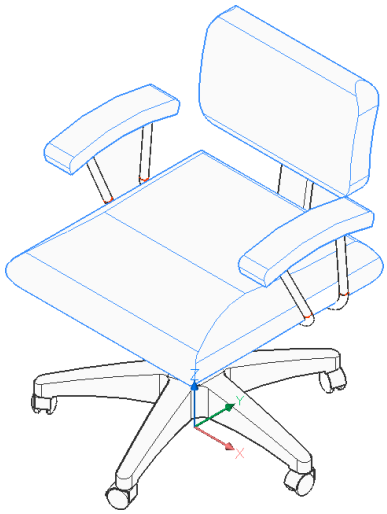
Rename
Allows you to rename custom visual styles.

Note: Default visual styles cannot be renamed.

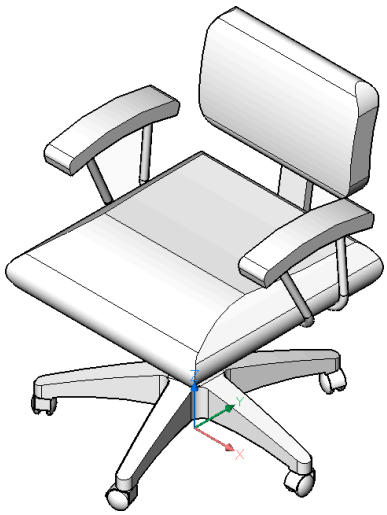
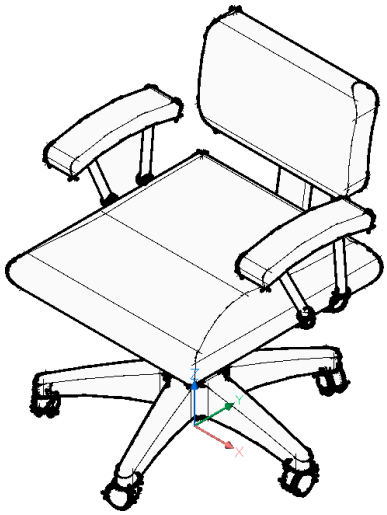
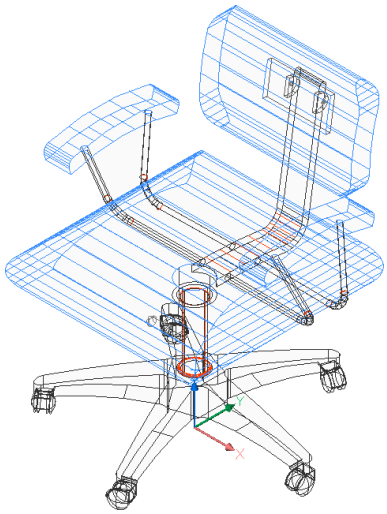
Delete
Deletes a visual style by name.

Note: The currently-used style and those defined by the program cannot be deleted.

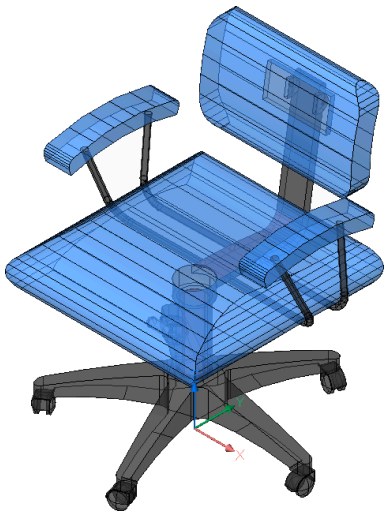
?
Lists the names of visual styles available in the drawing:

- 2dWireframe
- 
- Conceptual
- 
- Hidden
- 

- Realistic
- 
- Shaded
- 
- Shaded with edges
- 

- Shades of Gray
- 
- Sketchy
- 
- Wireframe
- 

- X-Ray
-



27.34 VLIDE command

Opens the BricsCAD LISP Advanced Development Environment (BLADE).

Shape Lite Pro Mechanical BIM

27.34.1 Description

Opens the BricsCAD LISP Advanced Development Environment (BLADE) to interactively edit and debug LISP applications. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

27.35 VLISP command

Opens the BricsCAD LISP Advanced Development Environment (BLADE).

Shape Lite Pro Mechanical BIM

27.35.1 Description

Opens the BricsCAD LISP Advanced Development Environment (BLADE) to interactively edit and debug LISP applications. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

27.36 VMLOUT command

Exports drawings in VML format, embedded in an HTML file (short for "vector markup language").

Shape Lite Pro Mechanical BIM

27.36.1 Description

Opens the Create HTML file dialog box to save data from the current drawing to an HTM file. The data is stored in VML format and embedded in an HTML file. You can view the file in a Web browser. However, you may need to install a VML plugin.

The output is in HTML format and looks something like this:

```
<html xmlns:v="urn:schemas-microsoft-com:vml">
<head>
  <object id="VMLRender" classid="CLSID:10072CEC-8CC1-11D1-986E-00A0C955B42E"></
object>
  <style>
    v\:* { behavior: url(#VMLRender); }
  </style>
</head>
<body>
 <v:group id="AN00001_" style="width:8in;height:8in;" coordSize="1600,1600">
 <v:shape style="width:1600;height:1600" path="nf m 214,42 l 213,41 214,41 e"/>
```

27.37 VPCLIP command

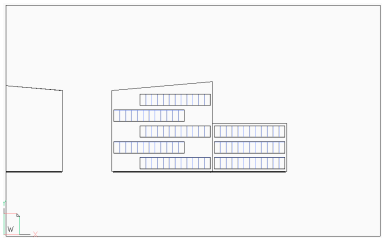
Clips a viewport to a specified clipping boundary.

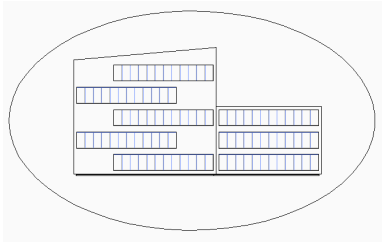
Shape Lite Pro Mechanical BIM

27.37.1 Description

Makes viewports from closed 2D entities (short for "viewport clipping") and can be easily used to show less of a drawing.

Note: This command only operates in the paper space of a layout tab and only works with already existing viewports. Use the MVIEW command to make additional viewports.





27.37.2 Method

Select a viewport to clip, then select the clipping entity.

Note:

- The clipping entity can be open or closed, made from polylines, splines or donuts. While the clipping entity does not need to be closed, it cannot cross over itself.
- The entity needs to be drawn in paper space.

27.37.3 Options within the command

Polygonal
Draws a polygonal clipping boundary made of straight and arc segments.

Draw Arcs
Draws an arc segment in the polygonal viewport. See the ARC command to explore the options within this option.

Close
Closes the polygon. The model space drawing is shown.

Distance
Specifies the distance and angle for the next polygonal segment.

Follow
Draws the next segment at the same angle as the previous polygonal segment. The length of the segment needs to be specified.

Undo
Undoes the last polygonal segment.

Note: Press Enter after the polygonal clipping boundary is drawn.

Delete
Deletes the clipping boundary and returns the original rectangular viewport without prompting or warning you.

Note:

- This option appears only when a clipped viewport is selected.
- The option can delete only the last clipping boundary applied to the viewport.

27.38 VPLAYER command

Changes the properties of layers in the current paper space viewport.

Shape Lite Pro Mechanical BIM

27.38.1 Description

Allows every viewport to show a different set of layers.

Note: This command does not operate in model space.

27.38.2 Options within the command

List frozen layers
Reports whether layers are frozen in the selected viewport. In the Command line, the frozen layers are shown after selecting a viewport.

Color
Overrides the color of specific layers in selected viewports. A new color is specified by entering the RGB code and applied to the layers in the selected viewports.

Red, Yellow, Green, Cyan, Blue, Magenta, White
Specifies the color for the layers in selected viewports.

Truecolor
Allows you to specify the true color for the layers in selected viewports, by entering the values for Red, Green, and Blue.

COlorbook
Opens a color book by entering its name and allows you to specify a color name from the loaded color book.

Note: The COLORBOOKPATH system variable specifies the folder(s) in which BricsCAD should look for color book files.

LineType
Overrides the linetype of specific layers in selected viewports.

LineWeight
Overrides the lineweight of specific layers in selected viewports.

TRansparency
Overrides the transparency of specific layers in selected viewports.

Freeze layers
Freezes specific layers in the current viewport.

Thaw layers
Thaws specific layers in the current viewport.

Reset layers
Resets frozen and thawed layers to their original settings.

New frozen layers
Creates new layers that are initially frozen when a new viewport is created.

Default visibility setting
Changes the default freeze/thaw setting for layers.

Frozen
Changes the layer's default property to frozen.

Thawed
Changes the layer's default property to thawed.

Specify viewport(s)
Specifies viewports to apply changes to.

All
Selects all viewports.

Select
Specifies a selection of viewports.

Current
Adapts changes to the current viewport.

eXcept current
Adapts changes to all viewports except the current viewport.

27.39 VPMAX command

Maximizes the current viewport.

Shape Lite Pro Mechanical BIM

27.39.1 Method

This command can be used both in Model space and Paper space to expand the selected viewport. When the Paper space only contains one viewport, running the command automatically selects the viewport. The command stops when there are no viewports available. If the selected object is not a viewport, the message **A single viewport is expected** appears in the Command line. The background color of the maximized viewport is automatically changed to the color of the Model space.

Note: Before using the VPMAX command in Model space, create viewports by using the VPORTS command.

Note: To minimize the viewport, use the VPMIN command.

27.40 VPMIN command

Minimizes the current viewport.

Shape Lite Pro Mechanical BIM

27.40.1 Method

This command can be used both in Model space and Paper space. This command restores the viewport to the form it was before it was maximized.

Note: To maximize the viewport use VPMAX command.

27.41 VPOINT command

Changes the 3D viewpoint.

Shape Lite Pro Mechanical BIM

Alias: -VIEWPOINT, -VP, -VPOINT, VIEWPOINT

27.41.1 Description

Changes the 3D viewpoint to change the view of the model through specifying several options.

Note: It is faster, and easier, to use the **LookFrom** widget to change the 3D viewpoint.

27.41.2 Method

Right-click in the drawing area to open the **Set Viewpoint** dialog box. For more information, see the related article **Set Viewpoint dialog box**.

Note: The dialog box opens only if perspective is off.

27.41.3 Options within the command

Set view point
Specifies the viewpoint by entering coordinates in the Command line or by specifying a point in the drawing.

Rotate
Changes the viewpoint by specifying angles.

Plan view
Displays the plan view of the current UCS. See the PLAN Command.

PErspective
Sets the perspective property of the viewport.

perspective ON
Turns on perspective viewing mode. Objects further away look smaller.

perspective OFf
Turns off perspective viewing mode, returning back to orthogonal viewing mode.

Front clipping
Sets the front clipping plane which obscures objects located between it and the camera.

frONt clipping on
Switch on front clipping at the current clipping distance.

front clipping OFf
Switch off front clipping.

Back clipping
Sets the back clipping plane which obscures objects behind it.

27.42 VPORTS command

Creates one or more viewports in model space.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: VIEWPORTS, VPORT, VW

27.42.1 Description

Creates one or more viewports in model space and allows you to see more than one view of the same drawing.

Note: To create viewports in paper space, use the MVIEW command.

27.42.2 Options within the command

?
Lists the names and x,y coordinates of saved viewports.

Note: Press F2 key to open the prompt history window.

Save
Saves the current viewport arrangement by name.

Note: If the name already exists, you are asked to overwrite the existing configuration or not.

Yes
Replaces the viewport configuration with a new one.

No
Saves the configuration with a different specified name.

Restore
Restores a named viewport configuration after entering the name of the configuration to restore.

Delete
Erases a named viewport configuration from the drawing.

Note: Only one configuration at the time can be deleted.

Single
Creates a single viewport of the current viewport, removing all others. This option can be used to return the drawing to its original single-viewport state.

Join
Joins two or more viewports to a single viewport after specifying the dominant viewport and the ones to join.

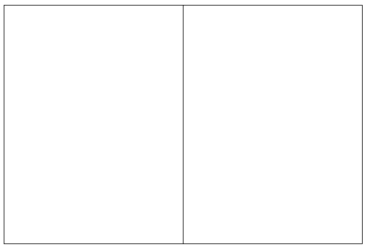
Note: If the two viewports were to join to form a non-rectangular shape, such as an L or a T shape, BricsCAD displays the message: 'The selected viewports do not form a rectangle.'. The 'Select viewport to join' prompts you to try again.

Create 2 viewports
Splits the current viewport into two rectangular viewports.

Horizontal
Creates two horizontal viewports, one above another.



Vertical
Creates two vertical viewports, one beside another.

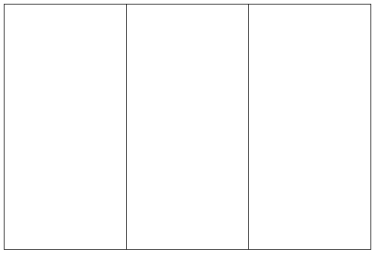


Create 3 viewports
Splits the current viewport into three rectangular viewports.

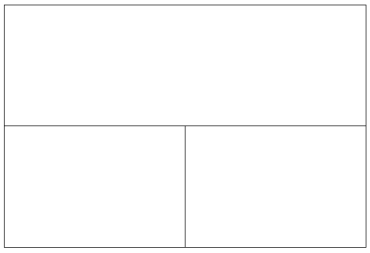
Horizontal
Creates three horizontal viewports, one above another.



Vertical
Creates three vertical viewports, one beside another.



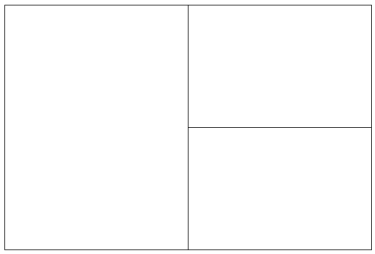
Above
Creates a single wide viewport above two side-by-side viewports.



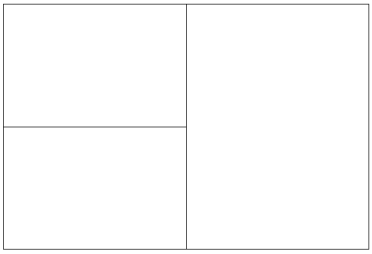
Below
Creates two side-by-side viewports above one wide viewport.



Left
Creates one tall viewport to the left of two stacked viewports.



Right
Creates one tall viewport to the right of two stacked viewports.



Create 4 viewports
Splits the current viewport into four rectangular viewports.



Note: This style is sometimes called "engineering" viewports, as it allows you to see the top, front, side, and isometric views at the same time, once you adjust the viewpoint in each viewport, which can be done with the MVSETUP command.

Note: Viewports can be resized by dragging their borders.

27.43 -VPORTS command

Creates one or more viewports in model space.

Shape Lite Pro Mechanical BIM

27.43.1 Description

Creates one or more viewports in model space and allows you to see more than one view of the same drawing.

For more information see VPORTS command.

27.44 VPSCALE command (Express Tools)

Displays the scale of a selected viewport.

Shape Lite Pro Mechanical BIM

Icon:

27.44.1 Method

The VPSCALE command only works in paper space.

27.45 VPSYNC command (Express Tools)

Aligns the view in layout viewports with a reference viewport.

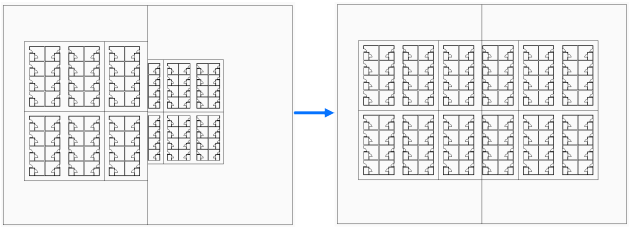
Shape Lite Pro Mechanical BIM

Icon:

27.45.1 Method

Select the reference viewport, then select viewports to aligned to the reference viewport.

Note: The VPSYNC command only works in paper space.



27.46 VSCURRENT command

Sets the visual style for the current viewport.

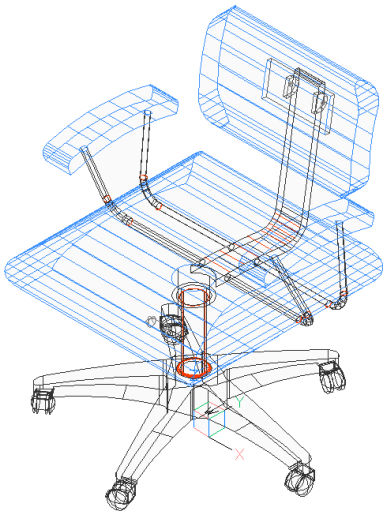
Shape Lite Pro Mechanical BIM

27.46.1 Description

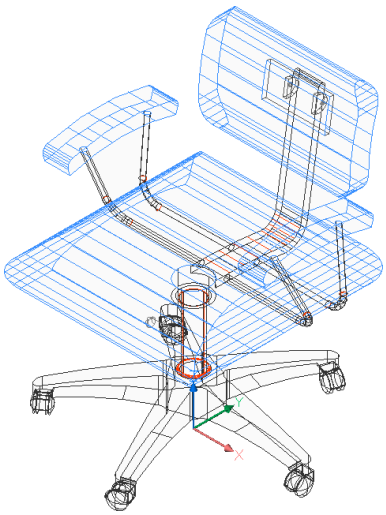
Sets the visual style for the current viewport in the Command line.

27.46.2 Options within the command

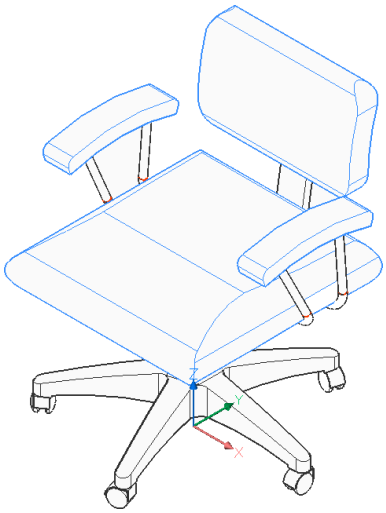
2dwireframe
Uses lines and curves to represent the entities' borders.



Wireframe
Uses lines and curves to represent the entities' borders.



Hidden
Uses lines and curves to represent entities' borders, but hiding the faces that are not visible.



Realistic

Smooths the boundaries between the faces of the polygons and shades the entities. Materials attached to the entities are shown.



Conceptual

Although the effect is less realistic, it can help you notice the entities' details more clearly.



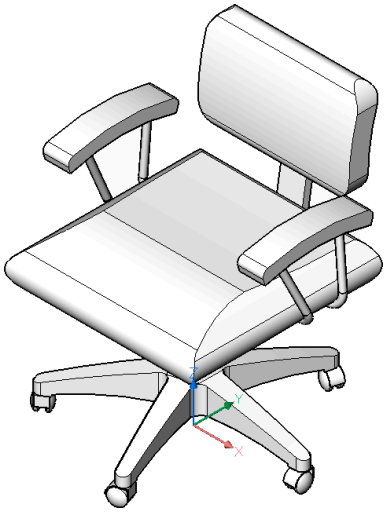
Shaded
The entities are smoothly shaded.



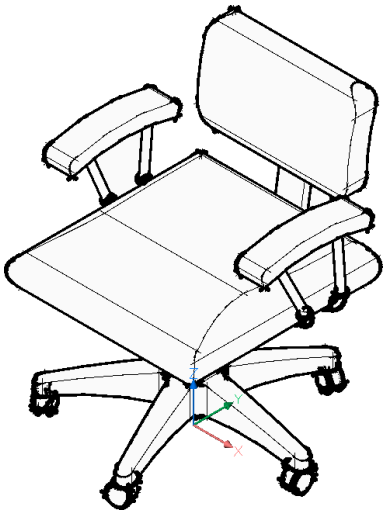
shaded with Edges
The entities are smoothly shaded, but they also have the edges highlighted.



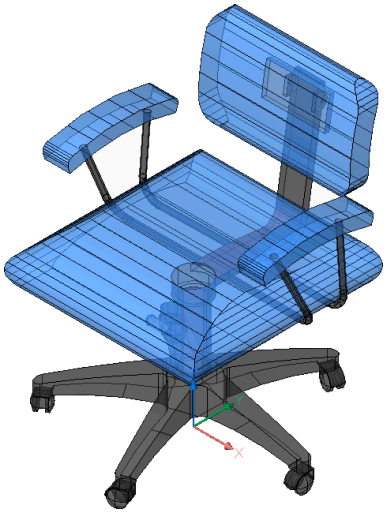
shades of Grey
The entities are smoothly shaded with monochromatic, grey tones.



SKetchy
Uses only black and white, making the models look hand-drawn.



X-ray
Changes the opacity of the models, making them seem transparent.



Other
Selects a user-defined visual style that was created with the VISUALSTYLES command.

cUrrent
Specifies the current visual style and gives the opportunity to keep it.

Note: The styles named in this list are the ones included with the program.

27.47 VSLIDE command

Opens the **View snapshot** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

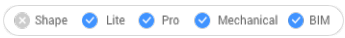
Alias: VS, VSNAPSHOT

27.47.1 Description

Opens the **View snapshot** dialog box to select an SLD, SLB, EMF, or WMF file to view in the current viewport. You can use the REDRAW command to remove the slide.

27.48 VTOPTIONS command

Opens the Settings dialog box with the **View Transition Options** category expanded.



27.48.1 Description

Opens the **Settings** dialog box with the **View Transition Options** category expanded to view and modify relevant system variables.

28. W

28.1 WBLOCK command

Opens the **Write block to file** dialog box.

Shape Lite Pro Mechanical BIM

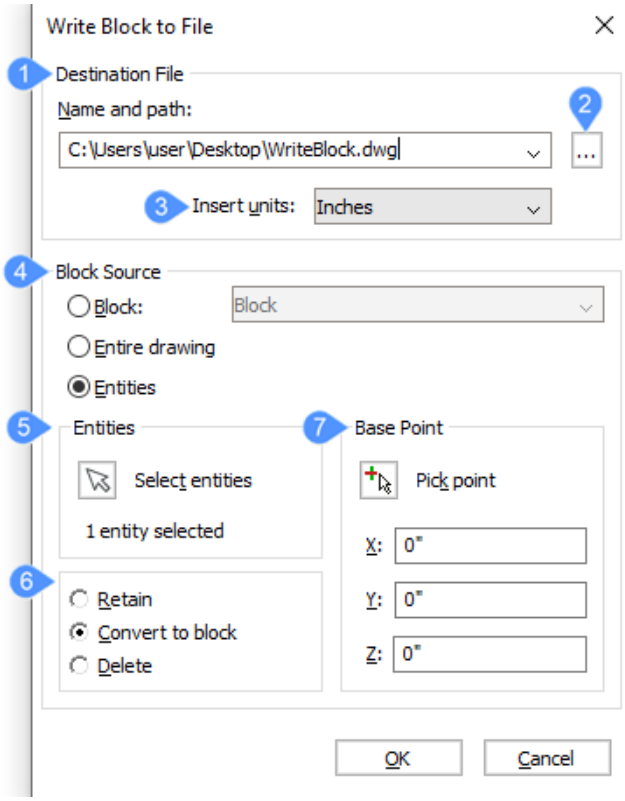
Icon:

Alias: W

28.1.1 Description

Opens the **Write block to file** dialog box to create a block definition as an external DWG file.

Note: Parametric operations are saved when creating external DWG files from parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command in the new file to make the parametric operations visible in the model space.



1 Destination file

2 Save Block

3 Insert units

4 Block Source

5 Entities

6 Options

7 Base Point

28.1.2 Destination file

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).

28.1.3 Save Block

Changes the path and file name. Opens the **Save Block** dialog box.

28.1.4 Insert units

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.

28.1.5 Block Source

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.

28.1.6 Entities

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.

28.1.7 Options

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.

  Note: If the **Delete** option was set, the OOPS command brings back the erased block entities.

28.1.8 Base Point

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.

28.2 -WBLOCK command

Saves blocks and other drawing entities to DWG/DXF files.

Shape Lite Pro Mechanical BIM

28.2.1 Description

Saves blocks and other drawing entities through the **Save Block** dialog box.

28.2.2 Options within the command

Block to save as WBlock
Specifies name of the block to write to a file.

Select entities
Selects entities to write to the file on disk.

Insertion point
Specifies the origin point of the drawing.

Select entities
Specifies the entities to create a block.

Note: The selected entities are deleted from the current drawing. To restore them, use the OOPS command.

Annotative
Saves the block as an annotative block.

& to include multiple blocks
Saves multiple blocks from this drawing to the file on disk.

Note: The definitions are saved to the new drawing.

*** for whole drawing**
Saves the whole drawing, equivalent to SAVEAS command.

28.3 WCASCADE command

Arranges windows.

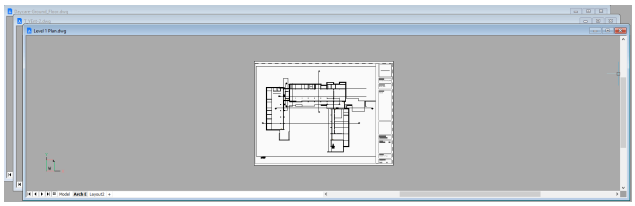
Shape Lite Pro Mechanical BIM

Icon: 

Note: This is a Windows only command.

28.3.1 Description

Overlaps all windows in a cascading fashion, with the current drawing the topmost one (short for “window cascade”).



28.4 WCLOSE command

Closes the current drawing.

Shape Lite Pro Mechanical BIM

28.4.1 Description

Closes the current drawing after it has been saved. If changes have been made since the last save, a BricsCAD dialog box offers you the opportunity to save the drawing before closing it.

28.5 WCLOSEALL command

Closes all drawings.

Shape Lite Pro Mechanical BIM

Alias: CLOSEALL

28.5.1 Description

Closes all drawings after they have been saved. If changes have been made to any drawings since their last save, a BricsCAD dialog box offers you the opportunity to save each of those drawings before closing them.

28.6 WEBLIGHT command

Creates web lights.

Shape Lite Pro Mechanical BIM

Icon:

28.6.1 Description

Creates web lights, which illuminate scenes with a 3D representation of the light's intensity distribution.

Note: Web lights are defined by IES files, which are provided by lighting manufacturers.

Note: The command is not allowed when LIGHTINGUNITS system variable = 0.

Note: If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), you need to specify the location of the light in 3D space and of the point at which the light is focused.

28.6.2 Options within the command

Name
Specifies a name for the light.

Intensity factor
Specifies the intensity of the light. Enter an intensity value between 0.00 and the maximum value supported by your system (max float = maximum value of a floating point number).

Status
Toggles the status of the light.

ON
Light is included when calculating renderings.

OFf
Light is excluded when calculating renderings.

Photometry
Specifies the photometric properties of the light:

Intensity (Cd)
Indicates the intensity expressed in candela (Cd).

Flux (Lm)
Indicates the luminous flux expressed in lumens (Lm).

Illuminance (Lx)
Indicates the illuminance expressed in lux (Lx) or in **Distance**, expressed in drawing units.

Color
Specifies a color name or choose an option.

?
Allows you to enter a color name.

*
Displays all the available color names.

Kelvin
Specifies a color temperature expressed in Kelvin degrees.

weB
Specifies the light distribution map, called **web** as an uneven light distribution can look like a spider's web:



File
Specifies a Web file name (.IES file). This is a text file that describes the intensity of a light source at many points on a spherical grid (image credit Vertheim).

X
Defines the X rotation for the web.

Y
Defines the Y rotation for the web.

Z
Defines the Z rotation for the web.

shadoW
Specifies the look of shadows cast by this light:

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges. Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources:

Shape
Defines the shape of the shadow circular (Disk) or rectangular (Rect) and their dimensions.

sAmples
Defines the sampling size of the shadow; larger numbers are more accurate but take longer to render.

Visible
Defines the shape visibility (type Yes or No to cast a shadow that represents shapes (more accurate) or that is rectangular (faster to render).)

filterColor
Specifies the color of the light by entering the true color (R, G, B) or entering an option.

True color (R, G, B)
The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values).

Index color
Specifies a color name or number.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness

Colorbook
Specifies the colorbook name and the color name inside of it. Verify if the colorbook (.acb) file is located in the folder indicated by the COLORBOOKPATH system variable.

28.7 WEDGE command

Creates a 3D Solid in the shape of a wedge.

Shape Lite Pro Mechanical BIM

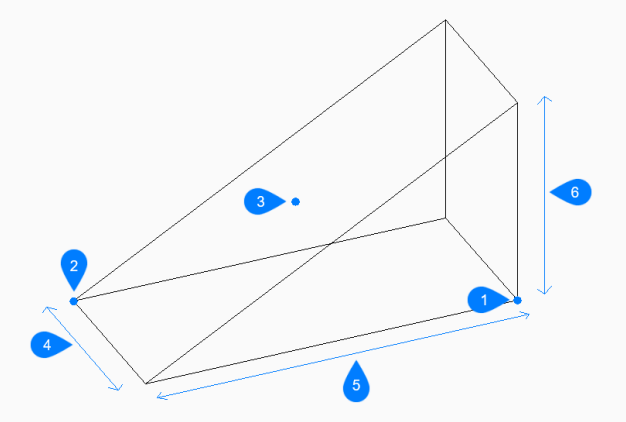
Icon:

Alias: WE

Note: In BricsCAD Lite, which doesn't support 3D Solids, the WEDGE command launches the AI_WEDGE command.

28.7.1 Description

Creates a 3D solid in the shape of a rectangular or square wedge. Choose from a combination of options including corner, center, length, height, and cube.



1 First corner

2 Opposite corner

3 Center of wedge

4 Width

5 Length

6 Height

28.7.2 Method

This command has 2 methods to begin creating a wedge:

- Set corner of wedge
- Center

28.7.3 Options within the command

Set corner of wedge
Allows you to begin creating a wedge by specifying a corner for the base of the wedge.

Set opposite corner
Specifies the opposite corner for the base of the wedge to apply both the length and width. The wedge is created parallel to the x and y axes.

Height of wedge
Specifies the height of the wedge. The tall end of the wedge is drawn from the first corner that you specified.

2Point
Specifies the height of the wedge as the distance between any two points.

Center
Begins creating a wedge by specifying the center of the wedge.

Set corner of wedge
Specifies a corner on the midplane of the wedge to apply both the length and width of the wedge's base. The wedge is created parallel to the x and y axes.

Cube
Specifies a single distance to use for the length, width, and height of the wedge.

Length of side
Specifies the length and angle for one side of the wedge.

Width of wedge
Specifies the width of the wedge.

28.8 WHOHAS command

Opens the **Open drawing** dialog box.

Shape Lite Pro Mechanical BIM

28.8.1 Description

Opens the **Open drawing** dialog box to select a DWG file to find out who has it open. After selecting the file and choosing **Open**, a **BricsCAD** dialog box displays. It indicates who has the file open and when they opened it.

28.9 WHTILE command

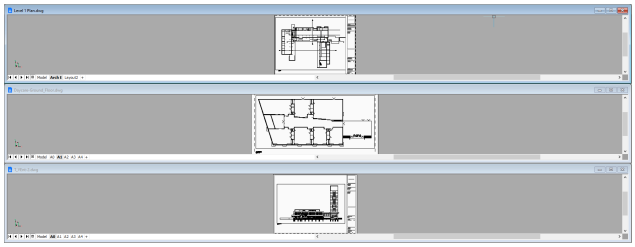
Tiles windows horizontally.

Shape Lite Pro Mechanical BIM

Icon:

28.9.1 Description

Tiles the drawing windows horizontally to maximize their width (short for "window horizontal tile").



28.10 WIARRANGE command

Lines up iconized drawing windows.

Shape Lite Pro Mechanical BIM

28.10.1 Description

Arranges the icons of minimized windows at the bottom of the graphical screen.

Note: This command works only when windows are minimized as icons.

- Command only available on the Windows platform.

28.11 WIPEOUT command

Creates a wipeout.

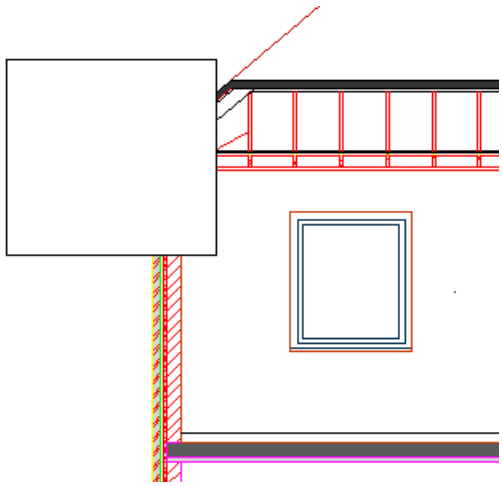
Shape Lite Pro Mechanical BIM

Icon:



28.11.1 Description

Creates a wipeout to mask parts a drawing. Options allow you to convert a polyline to a wipeout and control the display of wipeout frames.



28.11.2 Method

This command has 2 methods to begin creating a wipeout:

- Specify start point
- Polyline

28.11.3 Options within the command

Specify start point

Allows you to begin creating a wipeout by specifying a start point.

Specify next point
Specifies the next vertex of the wipeout.
You can continue adding vertices until you press **Enter** key to end the command.

Polyline
Creates a wipeout by selecting an existing closed polyline.

Delete selected polyline?
Allows you to choose whether to erase or retain the original polyline.

- **Yes**: erases the polyline.
- **No**: retains the polyline in addition to the wipeout.

Frames
Specifies whether to display wipeout frames. This is saved in the WIPEOUTFRAME system variable and applies to all wipeout entities in the drawing.

- **On**: displays and plot wipeout frames.
- **Off**: hides wipeout frames.
- **Display but not plot**: displays but does not plot wipeout frames.

Undo
Undo the last wipeout vertex and continue drawing from the previous one.

Close
Automatically draw a wipeout segment from the last vertex to the first one. This ends the command.

28.12 WMFIN command

Imports data from a WMF/EMF file into the current drawing.

Shape Lite Pro Mechanical BIM

Alias: WI

28.12.1 Method

Opens the **Select WMF/EMF file** dialog box to import data from the selected file into the current drawing.

28.12.2 Options within the command

Specify insertion point
Allows you to choose the insertion point. You can also define it by picking the point in the drawing area.

Specify X scale
Allows you to set the X scale value.

Specify Y scale
Allows you to set the Y scale value.

Specify rotation
Allows you to set Z rotation value. You can also define it by picking the angle in the drawing area.

28.13 WMFOUT command

Saves data from the current drawing to a WMF/SLD/EMF file.

Shape Lite Pro Mechanical BIM

Alias: WO

28.13.1 Method

Opens the **Create WMF file** dialog box to save data from the current drawing to the selected file format.

Select the entities you want to export.

28.14 WNEXT command

Switches to the next window.

Shape Lite Pro Mechanical BIM

28.14.1 Description

Brings another drawing window to the foreground (short for "window next"). It is useful when multiple drawings are open. Meant for use by macros.

The program changes focus to the next window. Windows are switched to in the order that they were created.

28.15 WORKPLANEACTIVATE command (Experimental)

Activates a Work Plane.

Shape Lite Pro Mechanical BIM

Alias: ACTIVATEWORKPLANE

28.15.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.15.2 Method

Select a Work Plane in the drawing or type its name in the command line to activate it so you can draw a sketch (2D entities) on it.

Note: Work Planes can be created using the WORKPLANECREATE command.

28.15.3 Options within the command

Deactivate

Deactivates the Work Plane.

Note: A warning message appears if no Work Plane is active.

Name
Specifies the name of the Work Plane to be activated. Note: A warning message appears when attempting to activate another Work Plane while one is already active.

28.16 WORKPLANECREATE command (Experimental)

Creates a Work Plane.

Shape Lite Pro Mechanical BIM

Alias: CREATEWORKPLANE

28.16.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future.

Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.16.2 Description

Creates a Work Plane that can be used to create a sketch.

Note: Work Planes have a visible representation in the Model Space and are available in the **Mechanical Browser** panel.

Right-click a Work Plane in the Mechanical Browser panel to open a context menu. The context menu allows you to create a sketch on the Work Plane. Also, you can Zoom to or Select the Work Plane.

28.16.3 Options within the command

All
Creates all XY, YZ, and XZ Work Planes at once.

XY
Creates an XY Work Plane.

YZ
Creates a YZ Work Plane.

XZ
Creates an XZ Work Plane.

Face
Creates a Work Plane on a selected flat face.

Offset
Creates a Work Plane parallel to a selected Work Plane or a flat face to a specified distance.

Note: You can edit the offset distance of the Work Plane from the **Mechanical Browser** panel.

Name
Allows you to enter the name of the reference Work Plane.

28.17 WORKPLANEHIDE command (Experimental)

Hides a Work Plane.

Shape Lite Pro Mechanical BIM

Alias: HIDEWORKPLANE

28.17.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.17.2 Method

Select a Work Plane in the drawing to hide it from the Model Space.

Note: The visibility of a Work Plane can also be controlled via the right-click menu in the **Mechanical Browser** panel.

28.17.3 Options within the command

All
Hides all Work Planes in the drawing.

28.18 WORKPLANELINK command (Experimental)

Links 2D entities with a Work Plane.

Shape Lite Pro Mechanical BIM

Alias: LINKWORKPLANE

28.18.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.18.2 Method

Select the sketch (2D entities) and the desired Work Plane that is available in the same spatial orientation to create a link. It creates an association between the sketch and the selected Work Plane to maintain their spatial relationship, allowing for consistent positioning and dynamic updates when parameters change.

Note: The entities must lie on the same plane and remain in the same spatial orientation.

28.18.3 Options within the command

Name
Enter the name of the Work Plane.

28.19 WORKPLANESHOW command (Experimental)

Shows a hidden Work Plane.

Shape Lite Pro Mechanical BIM

Alias: SHOWWORKPLANE

28.19.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.19.2 Method

Select a hidden Work Plane temporarily highlighted in the drawing to control it's visibility in the Model Space.

Note: The visibility of a Work Plane can also be controlled via the right-click menu in the **Mechanical Browser** panel.

28.19.3 Options within the command

All
Shows all Work Planes in the drawing.

28.20 WORKPLANEUNLINK command (Experimental)

Unlink 2D entities from the associated Work Plane.

Shape Lite Pro Mechanical BIM

Alias: UNLINKWORKPLANE

28.20.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

28.20.2 Method

Select the sketches (2D entities) to unlink from a Work Plane.

28.21 WORKSETS command

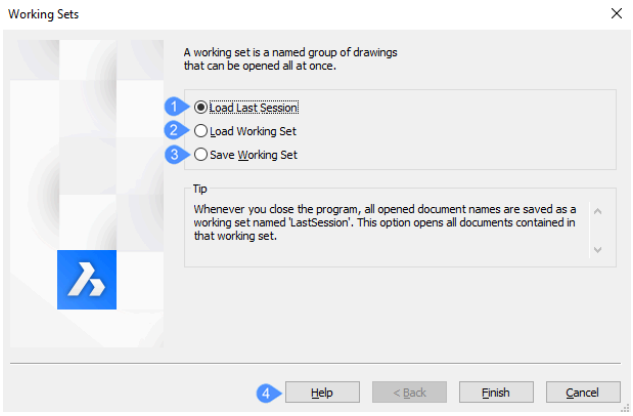
Opens the **Working sets** dialog box.

Shape Lite Pro Mechanical BIM

Icon:

28.21.1 Description

Opens the **Working sets** dialog box to create and load sets of drawing files. This makes it easy to load an entire group of drawings at once.



1 Load Last Session

2 Load Working Set

3 Save Working Set

4 Help

28.21.2 Load Last Session

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.

28.21.3 Load Working Set

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.

28.21.4 Save Working Set

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.

Note: Working sets are saved in the registry and so they are defined differently for each user profile. See the **PROFILEMANAGER command** article to learn more about user profiles.

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

28.21.5 Help

Opens the Bricsys Help article about WORKSETS command.

28.22 WORKSPACE command

Sets the current workspace.

Shape Lite Pro Mechanical BIM

Icon: 

28.22.1 Description

Sets the current workspace and lets you create, modify and save workspaces.

28.22.2 Options within the command

set Current
Sets the current workspace after entering a name.

?
Lists all the available workspaces.

Save As
Saves the current settings to a new workspace.

Rename
Renames a workspace.

Delete
Deletes a workspace.

SEttings
Opens the Workspaces tab of the **Customize** dialog box. Make adjustments to the workspace of your choice.

28.23 WPREV command

Displays the previous drawing window.

Shape Lite Pro Mechanical BIM

28.23.1 Description

Displays the previous drawing window when multiple drawings are open.

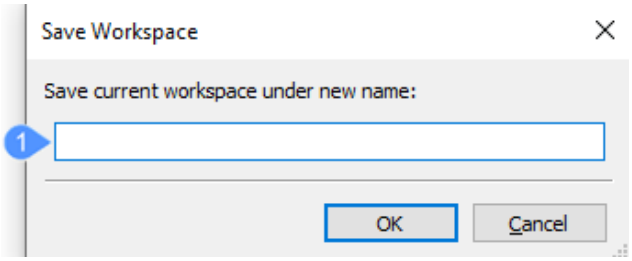
28.24 WSSAVE command

Opens the **Save workspace** dialog box.

Shape Lite Pro Mechanical BIM

28.24.1 Description

Opens the **Save workspace** dialog box to save the current workspace configuration to a new workspace name.



1 Save current workspace under new name

28.24.2 Save current workspace under new name

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.

28.25 WSSETTINGS command

Opens the **Customize** dialog box.

Shape Lite Pro Mechanical BIM

28.25.1 Description

Opens the **Customize** dialog box to customize the user interface.

28.26 WVTILE command

Tiles windows vertically.

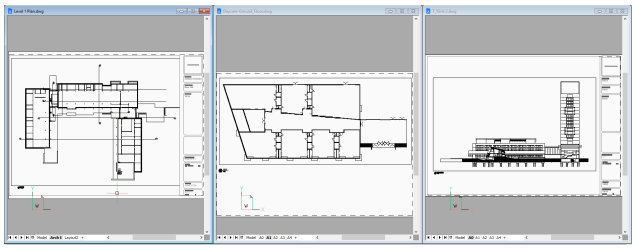
Shape Lite Pro Mechanical BIM

Icon:

28.26.1 Description

Tiles drawing windows vertically to maximize their height.

BricsCAD tiles all windows vertically, with the most recent drawing window highlighted.



29. X

29.1 XATTACH command

Attaches an external referenced file in the current drawing.

Shape Lite Pro Mechanical BIM

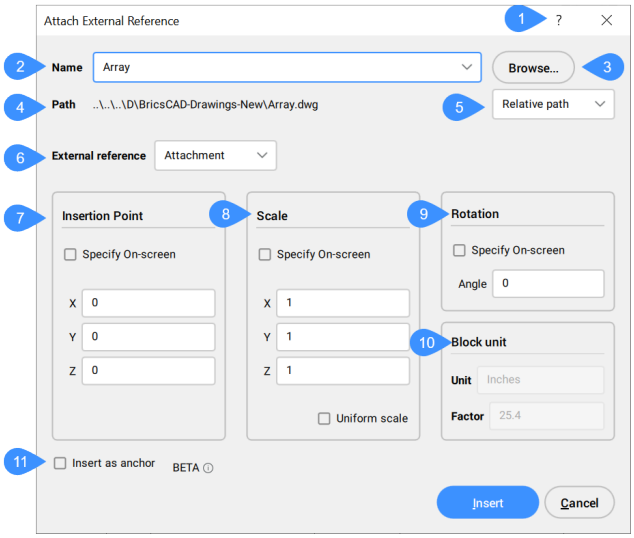
Icon:

Alias: XA

29.1.1 Description

Opens the **Select reference file** dialog box to select a DWG file to reference to in the current drawing. After selecting the file, click the **Open** button to display the **Attach external reference** dialog box. It enables you to specify where and how to attach the DWG file. The attached file will be visible in the **Attachments** panel.

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

11 Insert as anchor

29.1.2 Help

Opens the Bricsys Help article about the XATTACH command.

29.1.3 Name

Specifies the name of the DWG file attached to the current drawing.

29.1.4 Browse

Opens the **Select Reference File** dialog box to allow you to choose a different DWG file.

29.1.5 Path

Displays the path of the drawing file.

29.1.6 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**.

29.1.7 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.

29.1.8 Insertion Point

Specifies the location of the Xrefs lower-left corner:

Specify On-screen
Determines how to 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 Z coordinates for the Xrefs insertion point. Use 0,0,0 to insert the Xref at the drawing's origin.

29.1.9 Scale

Specifies the size of the Xref.

Specify On-screen
Determines how to 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 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.

29.1.10 Rotation

Specifies the rotation angle of the Xref.

Specify On-screen
Determines how to 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.

29.1.11 Block Unit

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

Unit
Reports the INSUNITS setting of the drawing being attached.

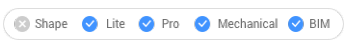
Factor
Reports the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.

29.1.12 Insert as anchor

Anchors an external reference to a face of a solid. This anchor links the external reference to the face. The external reference will stay on the face when the host solid is moved or edited.

29.2 -XATTACH command

Attaches an external referenced file in the current drawing via Command line.



29.2.1 Method

Specify the path to a DWG file, the insertion point, the scale factor and rotation angle to insert the external referenced file in the current drawing.

29.2.2 Options within command

Insertion point for block
Specifies the insertion point in the drawing.

Corner
Defines the X, Y, and Z scales at the same time, using the insertion point and the selected other corner for scale.

scale X Y Z
Specifies the X, Y, and Z scale factors of the Xref in the X, Y, and Z directions.

Scale
Specifies the size of the Xref.

X scale
Specifies the X scale factor of the Xref in the X direction.

Y scale
Specifies the Y scale factor of the Xref in the Y direction.

Z scale
Specifies the Z scale factor of the Xref in the Z direction.

Rotate
Specifies the rotation angle.

29.3 -XBIND command

Binds to the current drawing one or more definitions of named entities from an attached Xref via Command line.

Shape Lite Pro Mechanical BIM

29.3.1 Description

Binds definitions of named entities (symbols) from the attached Xref and allows you to use them as any other named entities in the current drawing.

29.3.2 Method

Choose which types of symbol to bind to the drawing and insert the dependent entity name in the Command line. You have to first specify the Xref name, then add the vertical bar character "|" and dependent symbol name.

Note: After the bind is done, the vertical bar character "|" from each Xref dependent entity is replaced with a number (usually 0) between two dollar signs ($).

29.3.3 Options within command

Blocks
Allows you to bind to the current drawing one or more blocks from the attached Xref.

Note: Any block, dimension style, layer, linetype, or text style that is referenced by entities in the block is also bound.

Dimension styles
Allows you to bind to the current drawing the dimension styles from the attached Xref.

LAyers
Allows you to bind to the current drawing the layers from the attached Xref.

Note: To bind a layer whose associated linetype is not CONTINUOUS, the command also binds the referenced linetype.

LineTypes
Allows you to bind to the current drawing the line types from the attached Xref.

text Styles
Allows you to bind to the current drawing the text styles from the attached Xref.

29.4 XCLIP command

Clips the display of a selected Xref or block reference to a specified clipping boundary.

Shape Lite Pro Mechanical BIM

Icon: 

29.4.1 Method

Choose one or more Xrefs or Block references to be clipped using the desired clipping boundary.

Note: Enter **ALL** in the **Select entities** option in the Command line to select all the Xrefs and Block references in the current drawing.

29.4.2 Options within the command

ON
Turns On clipping boundaries that were turned Off by the **OFF** option.
The area outside the clipping boundary is hidden from the view.

OFF
Turns Off clipping boundaries so that all of the selected Xrefs or Block references are fully visible again.
The clipping boundary is saved, until it is deleted, and it can be turned on again with the **ON** option.

Clipdepth
Sets the front and back clipping points only for 3D Xrefs or 3D Block references.

Specify front clip point
Specifies the position of the front clipping plane by picking the point or entering coordinates.

Specify back clip point
Specifies the position of the back clipping plane by picking the point or entering coordinates.

Distance
Specifies the position of the clipping points by entering the distance from the clipping boundary to the front or back clipping points.

Remove
Removes the front or back clipping point.

Note:

- This option does not operate until you create at least one clipping boundary.
- The cutting plane is parallel to the current clipping boundary.
- When picking the front or back clipping point you can define the cutting plane by creating a polyline through the points you want to cut.

Invert
Inverts the clipping mode so that the entities are clipped either outside or inside the specified boundary.

Note: The **Misc** section of the **Properties** panel also allows you to turn the clipping On and Off, and to invert it.

Delete
Erases the clipping boundaries from the selected entities without prompting or warning you.

generate Polyline
Draws a polyline over a selected clipping boundary allowing you to edit the clipping boundary, but in an indirect manner. This new generated polyline could be edited with PEDIT command and reused as a **New boundary** option.

New boundary
Allows you to add a clipping boundary to all selected entities.

Delete old boundary(s)?
This option appears in Command line only when an existing boundary is detected.

Yes
Deletes the previous boundary(s).

No
Exits the command.

CAUTION:

Select polyline
Allows you to choose any open or closed entity made from polylines, such as splines, polylines, rectangles, donuts, and polygons. While the polyline does not need to be closed, it cannot cross over itself.

Polygonal
Creates non-rectangular clipping boundaries, with a minimum of three sides.

Rectangular
Creates rectangular clipping boundaries.

29.5 XDATA command (Express Tools)

Attaches extended entity data (xdata) to a selected entity.

Shape Lite Pro Mechanical BIM

Icon:

29.5.1 Method

1 Select an entity.

2 Enter an application name. This will be the application ID.

Note: Use the XDLIST command to view the xdata attached to entities.

29.5.2 Options within the command

3Real
Allows you to specify 3 real numbers (group code 1010).

DIR
Allows you to specify a 3D World direction (group code 1013).

DISP
Allows you to specify a 3D World displacement (group code 1012).

DIST
Allows you to enter a distance (group code 1041).

Hand
Allows you to specify an entity handle (group code 1005).

Int
Allows you to specify a 16-bit integer (group code 1070).

LAyer
Allows you to specify a layer name (group code 1003).

LOng
Allows you to specify a 32-bit integer (group code 1071).

Pos
Allows you to specify a 3D World space position (group code 1011).

Real
Allows you to specify a real number (group code 1040).

SCale
Allows you to specify a scale factor (group code 1042).

STr
Allows you to specify an ASCII string (group code 1000).

Exit
Ends the XDATA command.

29.6 XDEDIT command (Express Tools)

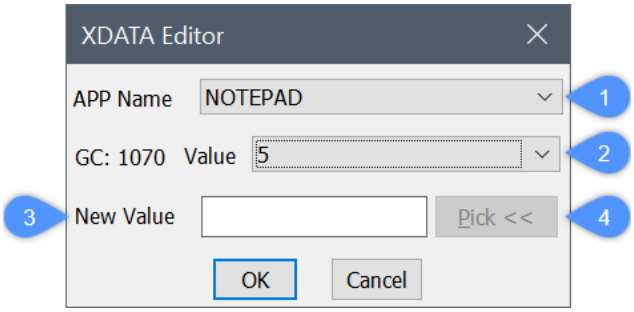
Edits the extended entity data (xdata) associated with an entity.

Shape Lite Pro Mechanical BIM

Icon:

29.6.1 Method

Opens the **XDATA Editor** dialog box which allows you to edit the extended entity data (xdata) associated with an entity.



1 APP Name

2 DXF Code

3 DXF Value

4 Pick <<

29.6.2 APP Name

Allows you to choose an application name from the list.

29.6.3 DXF Code

Allows you to change the XData DXF key code.

29.6.4 DXF Value

Allows you to specify a new DXF value.

29.6.5 Pick<<

Temporarily disables the **XDATA Editor** dialog box and allows you to pick a new value.

29.7 XDLIST command (Express Tools)

Lists the extended entity data (xdata) associated with an entity.

Shape Lite Pro Mechanical BIM

Icon:

29.7.1 Method

1 Select an entity.

2 Enter an application name.

29.7.2 Options within the command

*

Lists all applications.

29.8 XEDGES command

Extracts edges from 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

29.8.1 Description

Makes entities out of one or more edges found on 3D solids, 3D surfaces, and 2D regions; extracted edges become 2D entities in 3D space, such as arcs and lines.

Newly created entities are placed in the same location as the source edges, and on the current layer.

29.9 XFACES command

Extracts faces from 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

29.9.1 Description

Makes copies of one or more faces from one or more 3D solids and 3D surfaces.

29.9.2 Options within the command

Move

Move the extracted face/s by the distance indicated by two points or by a displacement vector.

29.10 XLINE command

Creates xlines.

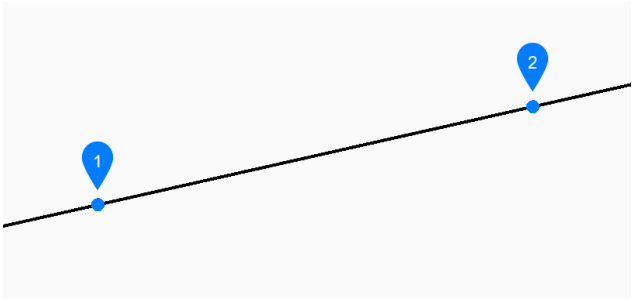
Shape Lite Pro Mechanical BIM

Icon: 

Alias: INFLINE, XL

29.10.1 Description

Creates an xline (infinitely long line) from a combination of options point, direction, and angle.



1 Point along line

2 Direction

29.10.2 Method

This command has 6 methods to begin creating an xline:

- Set point along line
- Horizontal
- Vertical
- Angle
- Bisect
- Parallel

29.10.3 Options within the command

Set point along line
Allows you to begin creating an xline by specifying a point on the xline.

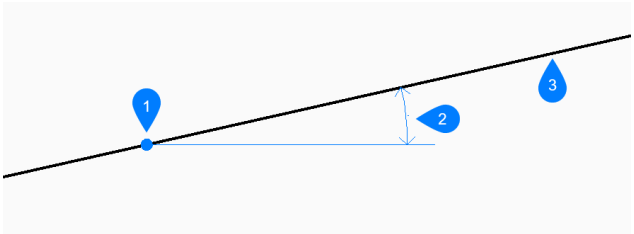
Direction
Specifies the direction of the xline from the starting point.

Note: You can continue adding unlimited xlines until you press **Enter** key to end the command.

Horizontal
Creates an xline parallel to the x-axis.

Vertical
Creates an xline parallel to the y-axis.

Angle
Creates a xline based on a specified angle.



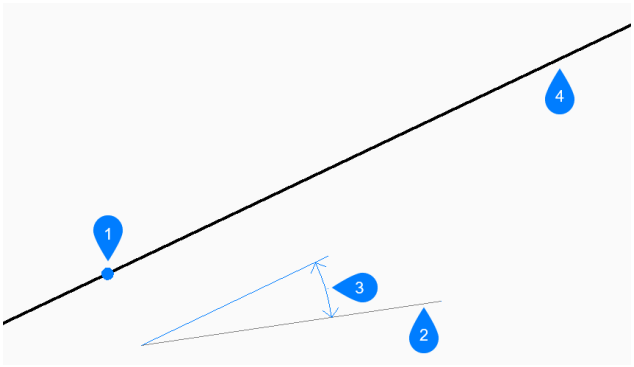
1 Location

2 Angle

3 Xline

Reference

Specifies the angle to place the xline relative to the selected entity.



1 Location

2 Reference entity

3 Angle

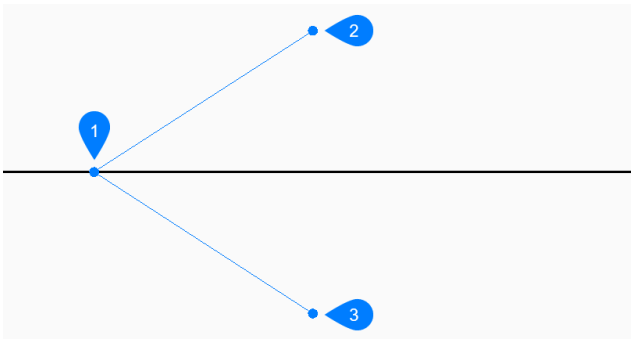
4 Xline

Location

Specifies the starting point of the horizontal, vertical, or angle xline.

Bisect

Creates an xline which bisects two imaginary lines.



1 Vertex point

2 Bisection angle start point

3 Bisection angle end point

Set vertex point

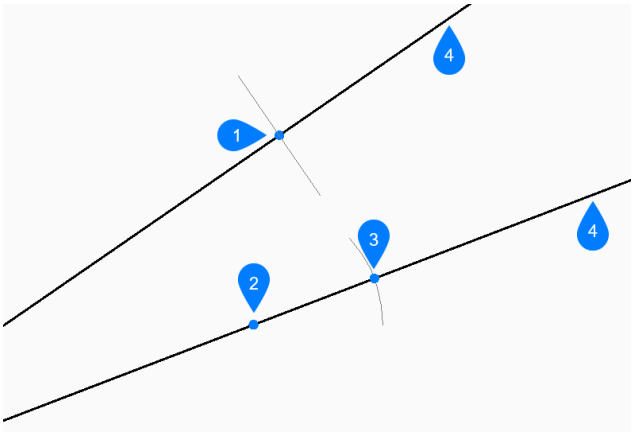
Specifies a point at the vertex of the two imaginary lines.

Entity

Allows you to select a line, arc or polyline segment to bisect.

When you select a line or polyline segment, the command draws the xline perpendicular to the segment's midpoint. When you select an arc or polyarc, the command draws the xline perpendicular to the arc's center and midpoint.

Note: This option works with splined polylines, but not with spline entities.



1 Perpendicular to line's midpoint

2 Perpendicular to arc's centerpoint

3 Perependicular to arc' midpoint

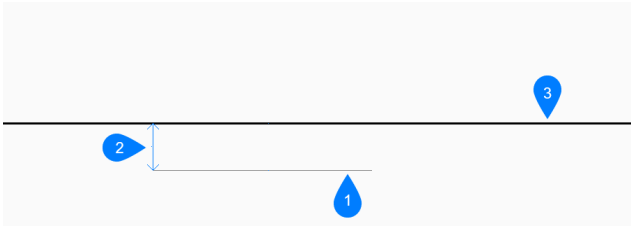
4 Xline

Parallel
Creates an xline parallel to a line or polyline segment.

Set offset distance for parallel infinite line
Specifies the offset distance for the xline.

Select entity for parallel infinite line
Allows you to select a line or polyline segment from which to offset the xline.

Side for parallel infinite line
Specifies the side on which to place the xline.

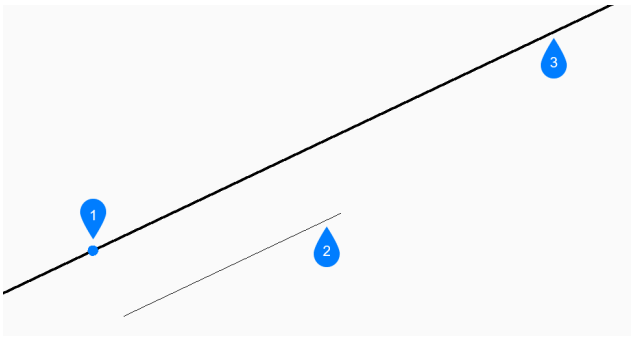


1 Line segment

2 Offset distance

3 Xline

Through point
Allows you to specify a point through which to draw the xline.



1 Through point

2 Line segment

3 Xline

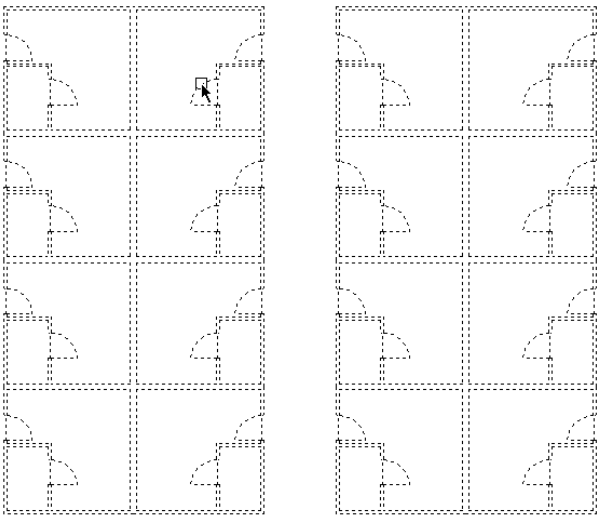
29.11 XLIST command (Express Tools)

Displays properties of entities nested in external references and blocks.

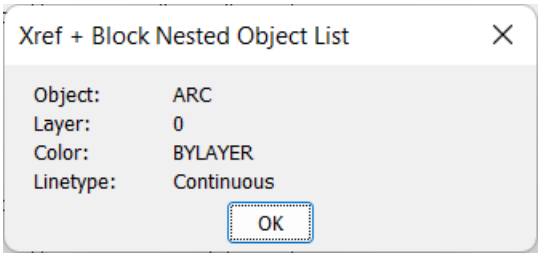
Icon:

29.11.1 Method

Select nested xref or block entity:



The **Xref + Block Nested Object List** dialog box opens and displays the properties of the selected nested entity.



29.12 -XLIST command (Express Tools)

Displays properties of entities nested in external references and blocks in the Command line.

Shape Lite Pro Mechanical BIM

29.12.1 Method

Select a nested xref or block entity. The properties of the selected nested entity are displayed on the Command line.

29.13 XOPEN command

Opens referenced drawings for editing.

Shape Lite Pro Mechanical BIM

Icon: 

29.13.1 Description

Opens an externally-referenced drawing (xref, attached to the current drawing) for editing in a separate drawing tab. When done editing, close the tab.

29.14 XPLODE command

Explodes entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: X

29.14.1 Description

Explodes entities providing control over the resulting entities.

Note: This command does not explode attached references, entities on frozen layers or basic entities such as lines, circles and arcs.

29.14.2 Options within the command

Entities to explode
Selects the entities to explode.

Note: Entities on frozen layers are not selected and not exploded.

Separately
Explodes one entity at a time.

All
Explodes all selected entities at once.

All listed
Allows to specify all properties for the selected entities.

Multiple
Specifies more than one property for the exploded entities by choosing properties one by one.

LAyer
Specifies the layer for the exploded entities

Note: The layer must already exist in the drawing.

Color
Specifies the color for the exploded entities.

LType
Specifies the linetype for the exploded entities.

ltScale
Specifies the linetype scale for the exploded entities.

Inherit all from parent
Inherits the color, layer and linetype scale from source entities.

29.15 XREF command

Opens the **Attachments** panel.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: XR, CLASSICXREF, ER, EXTERNALREFERENCES

29.15.1 Description

Opens the **Attachments** panel to display it in the current workspace. The **Attachments** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **Attachments** panel can be either floating, docked or stacked.

29.16 -XREF command

Attaches and detaches DWG files.

Shape Lite Pro Mechanical BIM

Alias: -XR

29.16.1 Description

Controls DWG drawing files that are attached to the current drawing (short for **external reference**).

29.16.2 Options within the command

? to list external references
Lists the names, paths and statuses of attached drawings, if any.

Attach
Opens the **Select File to Attach** dialog box to select a DWG file to attach. The Xref can be reloaded from the **Attachments** panel, and all its inserts including parametric ones are rebuilt.

Overlay
Opens the **Select File to Overlay** dialog box to select a DWG file to overlay.

Bind
Copies the contents of the attached drawing to the current drawing as a block reference. Parametric external references are converted to local parametric blocks.

Note: The BINDTYPE system variable defines how the Xref layers/objects will be brought in the current drawing.

Detach
Removes attached drawings.

Path
Changes the path (drive and folder) to the source DWG file, in case the original path no longer points to the file.

Reload
Reloads the attached drawing, after its content has changed.

Unload
Hides the attached drawing, but keeps it attached.

30. Y

31. Z

31.1 ZCENTER command

Toggles the Center of a Face 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.1.1 Description

Toggles the Center of a Face 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.2 ZINTERSECTION command

Toggles the Intersection 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.2.1 Description

Toggles the Intersection 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.3 ZKNOT command

Toggles the Knot 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon:

31.3.1 Description

Toggles the Knot 3D entity snap to enable or disable snapping to spline knot. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.4 ZMIDPOINT command

Toggles the Midpoint 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.4.1 Description

Toggles the Midpoint 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.5 ZNEAREST command

Toggles the Nearest 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.5.1 Description

Toggles the Nearest 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.6 ZNONE command

Disables all 3D entity snaps.

Shape Lite Pro Mechanical BIM

Icon: 

31.6.1 Description

Disables all 3D entity snaps to prevent the cursor from snapping to 3D entities. You can launch this command at the Command prompt to turn off running entity snaps. Doing so changes the value of the 3DOSMODE system variable to zero. You can also launch this command within another command to turn off entity snaps only for the current operation. This does not change the value of the 3DOSMODE system variable.

This command displays no prompts and has no options.

31.7 ZOOM command

Visually changes the size of the drawing within the current viewport.

Shape Lite Pro Mechanical BIM

Icon: 🔎

Alias: Z

31.7.1 Description

You can change the magnification of your drawing at any time by zooming.

31.7.2 Options within the command

In
Zooms in by 50%. It is equivalent to Zoom 2.

Out
Zooms out by 50%. It is equivalent to Zoom 0.5.

All
Displays the entire drawing.

Center
Zooms in or out about a specified center point.

Dynamic
Pans and zooms using a rectangular box which represents the viewport. This option first executes a zoom extents or zoom limits to show the original size of the viewport as a dashed rectangle.
Adjust the size of the view box dynamically, then pan it to the part of the drawing you want to view.

Extents
Displays the drawing to the extents of entities.

Note: Limits and frozen entities are ignored.

Left
Zooms in or out with the lower left corner specified by a point.

Previous
Displays the previous view, whether set by the Zoom, Pan, or View commands.

Right
Zooms in or out with the upper right corner specified by a point.

Scale
Specifies the absolute zoom scale factor.

Note: Less than 1 makes the drawing appear smaller. More than 1 makes the drawing appear larger.

nx
Specifies the zoom factor relative to the current zoom. For example, 2x makes the current view twice as large.

Nxp
Specifies the zoom factor in paper space relative to model space. For example, 2xp makes the model space view twice as large as the surrounding paper space.

Window
Zooms in to a rectangular area.

Object
Zooms in on a single entity or a selection set.

31.8 ZPERPENDICULAR command

Toggles the Perpendicular on a face 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon:

31.8.1 Description

Toggles the Perpendicular on a face 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.9 ZVERTEX command

Toggles the Vertex 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.9.1 Description

Toggles the Vertex 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.