BRICSCAD®

COMMAND REFERENCE

Updated for V18

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Command Prefixes

The following prefixes can be used with the BricsCAD commands:

Prefix	Type	Result
' (single quote or apostrophe)	Transparency	Allows a command to be executed inside another command. E.g. If you are drawing a line, type *'zoom* at the command prompt to temporarily interrupt 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 bar. E.G. Type *-layer* at the command prompt to suppress the Drawing Explorer / Layers dialog. All command options are available in the command bar. 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. E.g. if the ZOOM command is temporarily undefined '*.zoom* executes the ZOOM command transparently.

Command Prompt Format

Required input [FirstOption/SecondOption/ThirdOption]<what happens if you just press ENTER>:
e.g. the first prompt of the COPY command:
Enter base point [Displacement/mOde] <Displacement>:

Accessing the preference

command bar: *promptoptionformat*
settings dialog: *Program Options / Display / Prompt Option Format*

: ***promptoptionformat***
Prompts you in the command bar:

New value for PromptOptionFormat <0>: ***(Type 0, 1, 2, 3 or 4)***

Options

A command option has a **keyword**, a **description** and a **shortcut**. The shortcut is the keyword, without its lowercase characters.
e.g. the third option of the CIRCLE command:
Keyword = TanTanRad
Description = Tangent-Tangent-Radius
Shortcut = TTR

Option	Description
0	Description only; adjust casing to mark shortcut: Select center of circle or [2 Point/3 Point/Tangent-Tangent-Radius/turn Arc into circle/Multiple circles]: This is the default prompt option format in the English version. In other versions the default prompt option format depends on local standards.
1	Show keyword only: Select center of circle or [2Point/3Point/TanTanRad/Arc/Multiple]:
2	Show description and keyword in brackets: Select center of circle or [2 Point(2Point)/3 Point(3Point)/Tangent-Tangent-Radius(TanTanRad)/Turn arc into circle(Arc)/Multiple circles(Multiple)]:
3	Show Description and shortcut in brackets: Select center of circle or [2 Point(2P)/3 Point(3P)/Tangent-Tangent-Radius(TTR)/Turn arc into circle(A)/Multiple circles(M)]:
4	Show local keyword and global keyword in brackets; this is relevant in localized versions only. Select center of circle or [2Point/3Point/TanTanRad/Arc/Multiple]:

NOTE The PromptOptionTranslateKeywords user preference controls whether translations of command option keywords are loaded or not. If disabled, the local keyword will be a copy of the global (English) keyword. As a result global shortcuts can be used without an underscore.

?

BricsCAD (Windows): Opens the BricsCAD Help window.
BricsCAD (Linux): Opens the BricsCAD Online Help on the Bricsys website.
BricsCAD (Mac): Opens the BricsCAD Online Help on the Bricsys website.

Accessing the Command

command bar: *?*
transparent: *'?*
shortcut: *F1*
alias: *help*
menu bar: *Help | Help*
toolbar: *Standard |*

: ***?***
Displays the Help window.

See the Help command.

Command Options

See the Help command.

Related Commands

Help - opens the BricsCAD Help window.
HelpSearch - searches the BricsCAD Help and displays a list of matching topics in a dialog box.

2

2dIntersection

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

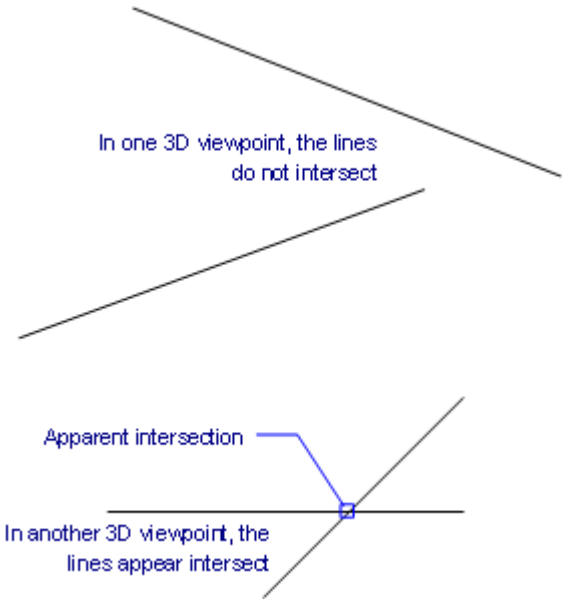
Accessing the Command

command bar: *2dintersection*
transparent: *2dintersection*
alias: *apparent*
toolbar: *Entity Snaps* |

: ***2dintersection***

Displays no prompt in the command bar.

This entity snap snaps to intersecting entities, including those that appear to intersect in 3D space:



When 2dintersection entity snap is on, the cursor snaps to the real and apparent intersections of entities.

Command Option

Option	Description
2dIntersection	Entering this command toggles apparent intersection entity snap. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Midpoint - toggles midpoint snap mode.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

3

3D

Draws 3D mesh objects: boxes, cones, cylinders, dishes, domes, meshes, pyramids, spheres, tori, and wedges (short for "three dimensional").

Accessing the Command

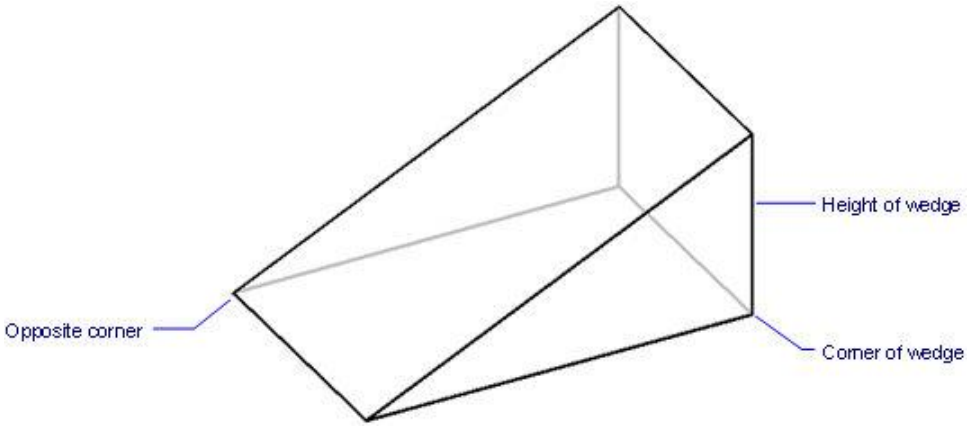
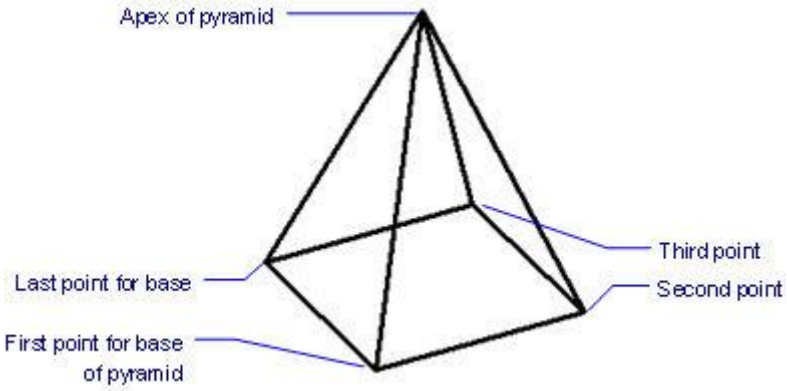
command bar: *3d*

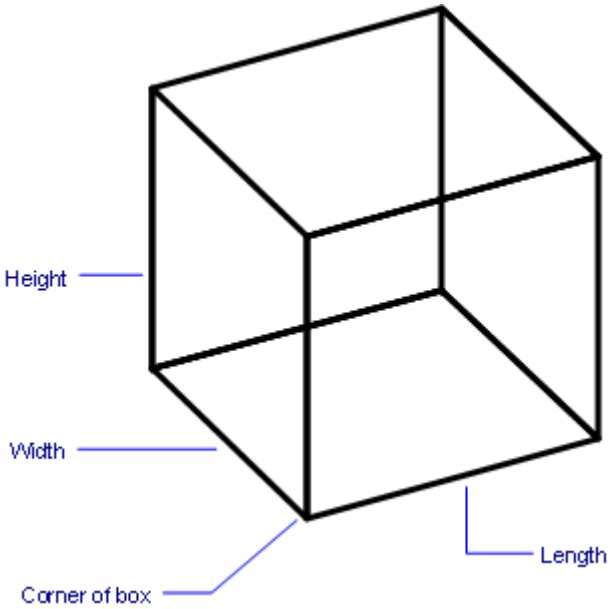
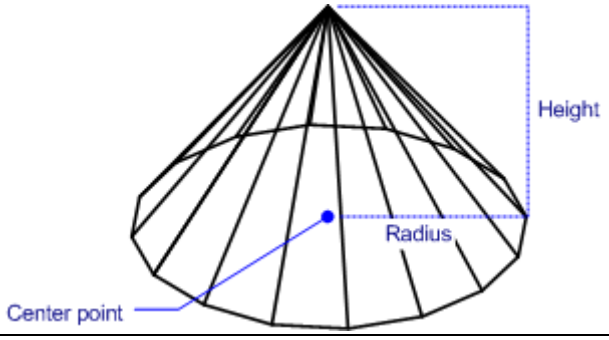
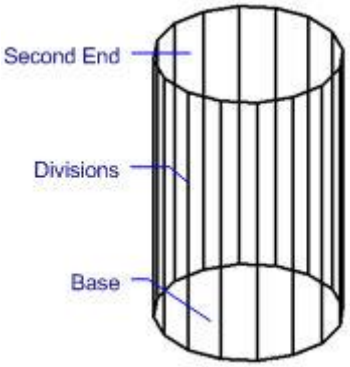
: ***3d***

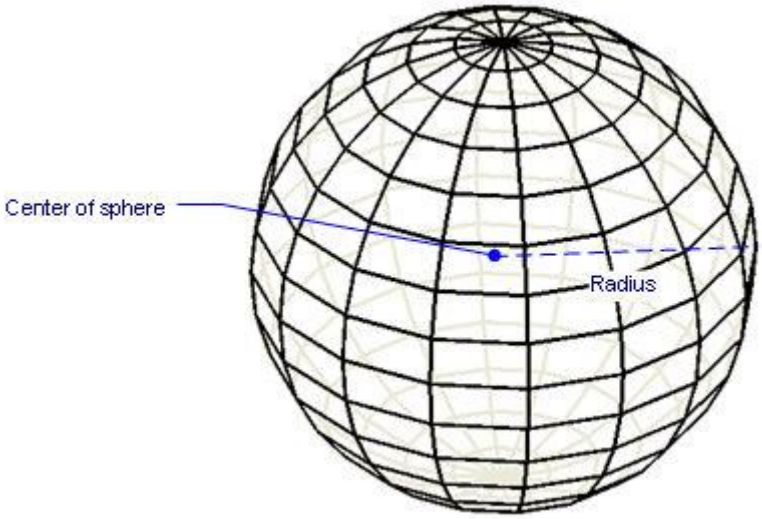
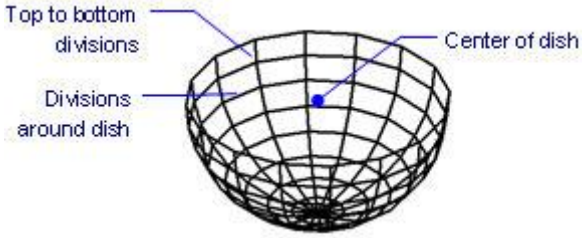
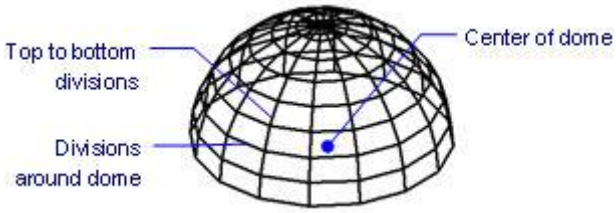
Prompts you in the command bar:
[**Wedge**/**Pyramid**/**Box**/**Cone**/**cyLinder**/**Sphere**/**DIsh**/**DOme**/**Torus**/**Mesh**]: *(Enter an option.)*

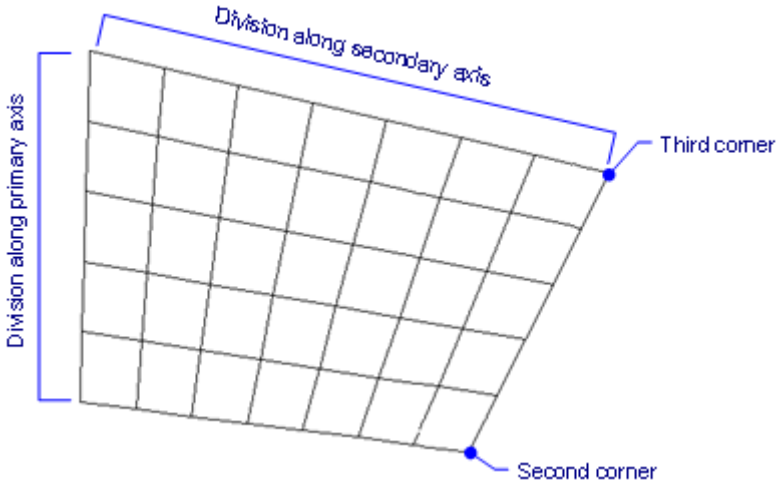
This command acts like a shell for the 3D mesh modeling commands.

Command Options

Option	Description
Wedge	Draws 3D wedges. See Ai_Wedge command. 
Pyramid	Draws 3D pyramids. See Ai_Pyramid command. 
Box	Draws 3D boxes. See Ai_Box command.

	
Cone	Draws 3D cones. See Ai_Cone command. 
cyLinder	Draws 3D cylinders See Ai_Cylinder command. 
Sphere	Draws 3D spheres. See Ai_Sphere command.

	
DIsh	Draws 3D dishes (half spheres). See Ai_Dish command. 
DOme	Draws 3D domes (half spheres). See Ai_Dome command. 
Torus	Draws 3D tori (donuts). See Ai_Torus command.

Mesh	Draws 3D meshes; prompts you:  First of four corners for **polygon** mesh - pick a point, or enter x,y,z coordinates. Second corner - pick a point, or enter x,y,z coordinates. Third corner - pick a point, or enter x,y,z coordinates. Last corner for mesh - pick a point, or enter x,y,z coordinates. Number of divisions along primary axis (between 2 and 256) - enter a number. Number of divisions along secondary axis (between 2 and 256) - enter a number.

Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.

Ai_Cone - draws 3D cones as mesh surfaces.

Ai_Cylinder - draws 3D cylinders as mesh surfaces.

Ai_Dish - draws 3D dishes as mesh surfaces.

Ai_Dome - draws 3D domes as mesh surfaces.

Ai_EdgeSurf and EdgeSurf - draws edge surfaces as mesh surfaces.

Ai_Pyramid - draws 3D pyramids as mesh surfaces.

Ai_RevSurf and RevSurf - draws revolved surfaces as mesh surfaces.

Ai_RuleSurf and RuleSurf - draws ruled surfaces as mesh surfaces.

Ai_Sphere - draws 3D spheres as mesh surfaces.

Ai_TabSurf and TabSurf - draws tabulated surfaces as mesh surfaces.

Ai_Torus - draws 3D donuts as mesh surfaces.

Ai_Wedge - draws 3D wedges as mesh surfaces.

3dCompare

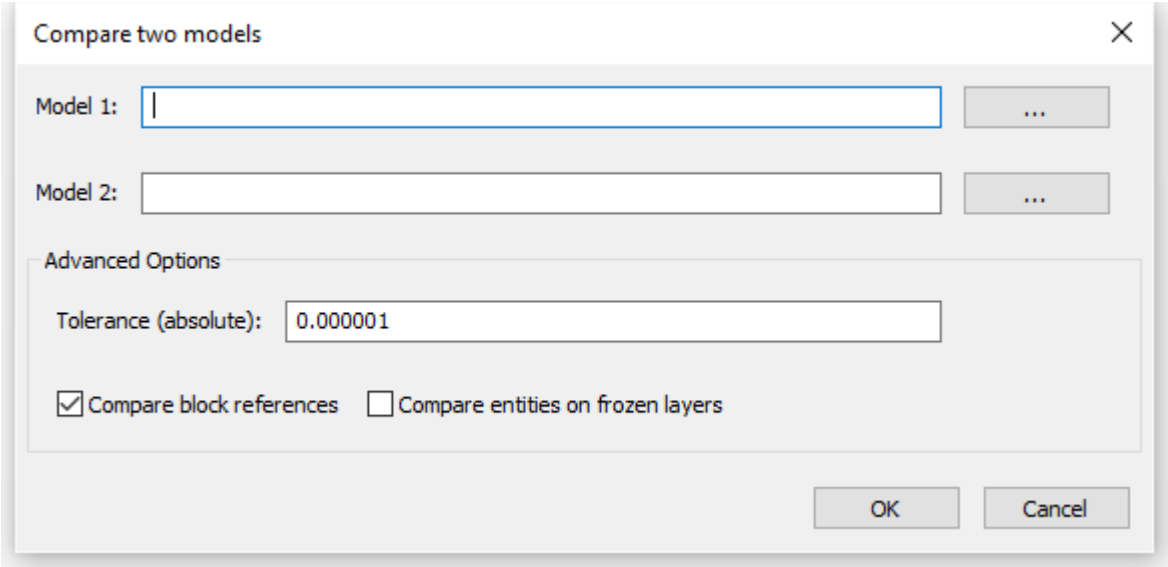
Platinum

Executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel.

Accessing the Command

command bar: *3dcompare*
toolbar: *3D Compare* |
ribbon: *tools* | *Compare 3D* |

: ***3dcompare***
Displays a dialog box:



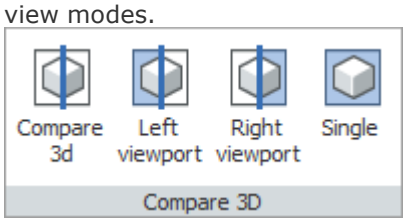
Do the following:

- Click the *Browse* button (...) of the *Model 1* and *Model 2* fields and select the two models you want to compare.
- Set the options.
- Click the *OK* button.

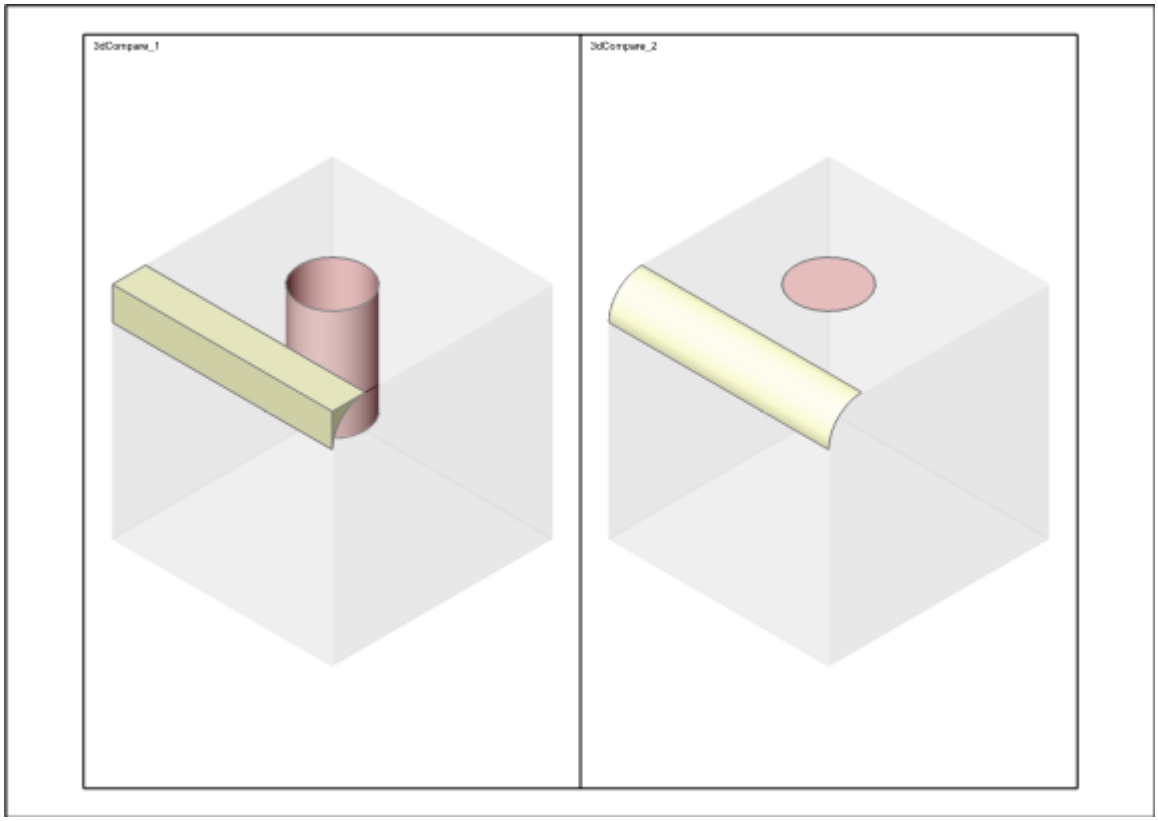
The selected files are attached to a new drawing named: Compare_<File_1>_<File_2>.dwg.

A layout named *Comparison* in this drawing contains two viewports, one for each model.

Differences between the models are reported in the Mechanical Browser panel.
The value of the 3DCOMPAREMODE system variable controls the display in the viewports of the *Comparison* layout.
The tools in the *Compare 3D* Ribbon panel of the *Tools* tab allow to switch between the various view modes.



Icon	Name	Description
	Left Viewport	Shows the differences in the left viewport of the *Comparison* layout.
	Right Viewport	Shows the differences in the right viewport of the *Comparison* layout.
	Single	Shows the differences in model space. When switching to this option, disable both layout viewport display options.



In the image above the differences display in both layout viewports.

Command Options

Option	Description
Tolerance	Threshold value for geometry comparison.
Compare block references	Specifies if the content of blocks should be compared. 3DCOMPARE can work with block structures of arbitrary depth which may contain solids in its nodes. Assemblies based on a block structure as well as regular blocks and inserts can be analyzed also.
Compare entities on frozen layers	Specifies if entities on frozen layers should be compared.

Related Commands

BmBrowser - toggles the visibility of the Mechanical Browser panel.
DwgCompare - compares a selected drawing with the current drawing during a comparison session.

3dArray

Constructs 3D static rectangular arrays and polar arrays (short for "three dimensional array").

Accessing the Command

command bar: *3darray*
aliases: *3a, array3d*
menu bar: *Modify | 3D Array*
toolbar: *Modify | Arrays |*

: ***3darray***
Prompts you in the command bar:

Select entities to array: **(Choose one or more entities.)**

Entities in set: 1

Select entities to array: **(Choose more entities, or press Enter to continue.)**

Type of array: Polar/<Rectangular>: **(Type P or R.)**

Rectangular array:

Number of rows in the array <1>: **(Enter a number, or press Enter to accept the default.)**

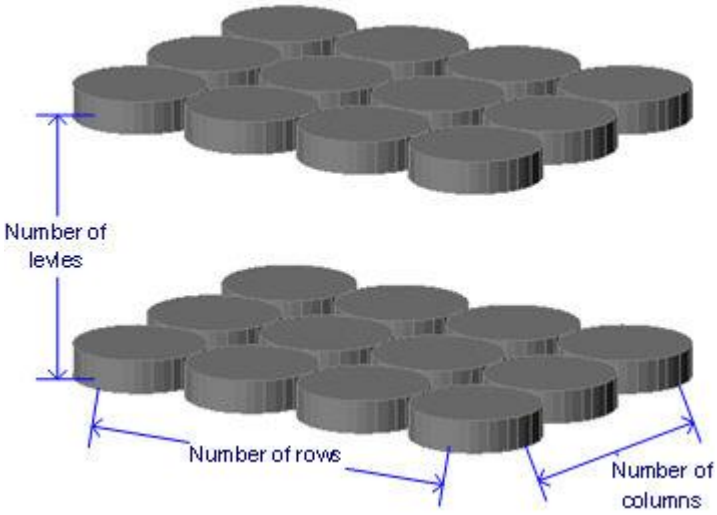
Number of columns <1>: **(Enter a number, or press Enter to accept the default.)**

Number of levels <1>: **(Enter a number, or press Enter to accept the default.)**

Vertical distance between rows: **(Enter a number, or press Enter to accept the default.)**

Horizontal distance between columns: **(Enter a number, or press Enter to accept the default.)**

Depth between levels: **(Enter a number, or press Enter to accept the default.)**



Polar array:

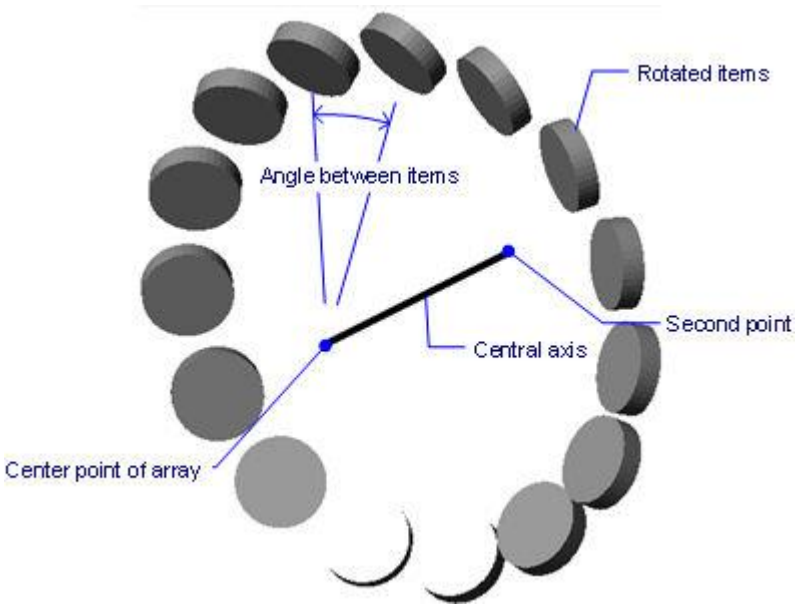
ENTER to specify angle between items/<Number of items to array>: **(Enter a number, or press Enter.)**

Angle to array (+ for ccw, - for cw) <360>: **(Enter a number, or press Enter to accept the default.)**

Rotate entities around the array? No/<Yes>: **(Type Y or N.)**

Center point of array: **(Pick a point, or enter x,y,z coordinates.)**

Specify second point along central axis of array: **(Pick a point, or enter x,y,z coordinates.)**



Command Options

Option	Description
Select entities to array	Specifies the entities to be arrayed. Use any entity selection method; press Enter when you are done.
Polar	Creates polar (circular) arrays.
Rectangular	Creates linear, rectangular, or square arrays.

Rectangular Array Options	Description
Number of rows in the array	Specifies the number of rows (x direction); for a linear array, enter 1. To draw the array downwards, enter a negative number.
Number of columns	Specifies the number of columns (y direction). To draw the array to the left, enter a negative number.
Number of levels	Specifies the number of levels (z direction). To draw the array downwards, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of entities.
Horizontal spacing between columns	Specifies the distance between columns of entities.
Depth between levels	Specifies the distance between levels of entities.

Polar Array Options	Description
ENTER to specify angle between items	Specifies the angle between each copy.
Number of items to array	Specifies the number of items in the array; enter a number larger than 1.

Angle to array	Determines the extent and direction of the array. • Enter **360** for a full circle of copies. • Enter a smaller number to draw a partial polar array. • Enter a positive angle to draw the array counterclockwise. • Enter a negative number to draw the array clockwise.
Rotate entities around the array	Determines whether copies are rotated when they are arrayed: • **No** does not rotate entities; all copies look the same as the original. • **Yes** rotates copies.
Center point of array	Specifies the starting point of the polar array's central axis. Pick a point, or enter x,y,z coordinates.
Specify second point along central axis of array	Specifies the endpoint of the polar array's central axis. Pick a point, or enter x,y,z coordinates.

Procedures

Creating a 3D rectangular array
Creating a 3D polar array

Related Commands

ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.

-Array - creates a static polar or rectangular array of entities; displays prompts at the command bar.

Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.

ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.

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

ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.

ArrayEdit - allows editing associative array entities and their source entities.

MInsert - inserts arrays of blocks.

3dConvert

Converts 3D solids to polyface meshes.

Accessing the Command

command bar: *3dconvert*

: ***3dconvert***
Prompts you in the command bar:

Select ACIS entities to convert: **(Choose one or more 3D solids.)**

Select ACIS entities to convert: **(Choose more 3D solids, or press Enter to continue.)**

Command Options

Option	Description
Select ACIS entities to convert	Chooses the 3D solids to be converted to polyface meshes.

Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

3dFace

Draws 3D 4-edged faces with optional invisible edges.

Accessing the Command

command bar: *3dface*
aliases: *3f, face*
menu bar: *Model | Meshes | 3D Face*
toolbar: *Meshes | 3D Faces |*

: ***3dface***
Prompts you in the command bar:

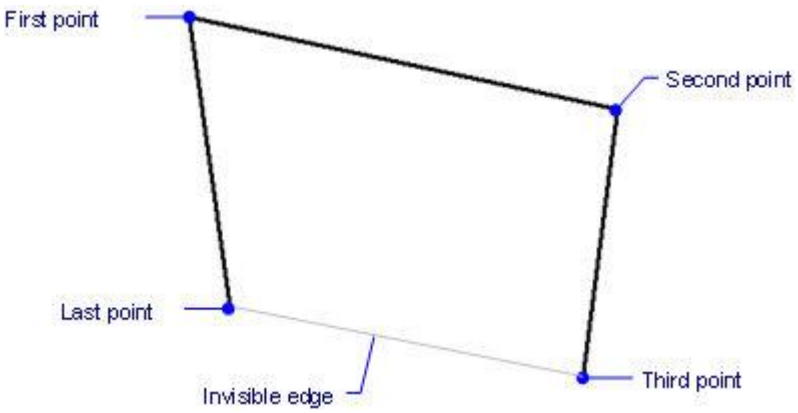
First point of 3D face [Invisible]: **(Pick a point, or type I.)**

Second point [Invisible]: **(Pick a point, or type I.)**

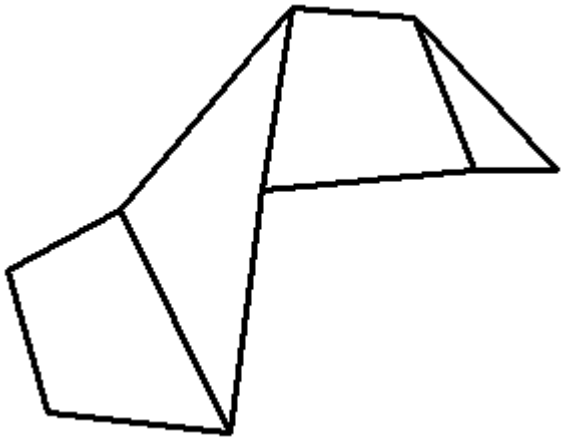
Third point of 3D face [Invisible]: **(Pick a point, or type I.)**

Last point [Invisible]: **(Pick a point, or type I.)**

Third point of 3D face [Invisible]: **(Pick a point, or type I, or press Enter to stop.)**



This command repeats the 'Invisible edge/<Third point of 3D face [Select Invisible edge(Invisible)]>' prompt so that adjoining faces can be drawn:



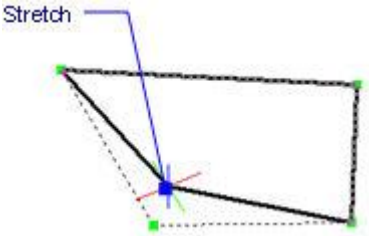
Command Options

Option	Description
First point of 3D face	Specifies the location of the first corner of the face. Pick a point, or enter x,y,z coordinates.
Second point	Specifies the location of the second corner of the face. Pick a point, or enter x,y,z coordinates.
Third point of 3D face	Specifies the location of the third corner of the face. Pick a point, or enter x,y,z coordinates.
Last point	Specifies the location of the fourth corner of the face. Pick a point, or enter x,y,z coordinates.
Invisible edge	Makes the next edge invisible; prompts you: **Third point of 3D face** - specify a point for the next vertex.

Grips Editing

3D faces can be edited directly through grips:

1. Select the 3D face. Notice that it has several grips.
2. Drag a grip to stretch the face.



Related Commands

3dMesh - draws 3D surface meshes of size m x n.
PEdit - edits 3D meshes.
PFace - draws 3D multi-sided meshes.

3dIntersection

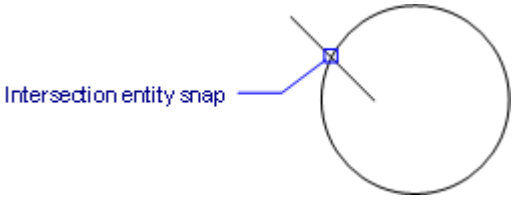
Toggles Intersection entity snap; snaps to the intersections of entities.

Accessing the Command

command bar: *3dintersection*
*t*ransparent: *3dintersection*
alias: *intersection*
toolbar: *Entity Snaps* |

: ***3dintersection***

Displays no prompts in the command bar.



This entity snap snaps to intersecting entities; to snap to apparent intersections, use 2dintersection entity snap.

Command Options

Option	Description
3dIntersection	Entering this command toggles Intersection entity snap. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

3dMesh

Draws 3D surface meshes.

Accessing the Command

command bar: *3dmesh*
alias: *mesh*
menu bar: *Model | Meshes | Polygon Mesh*
toolbar: *Meshes | 3D Face |*

: ***3dmesh***
Prompts you in the command bar:

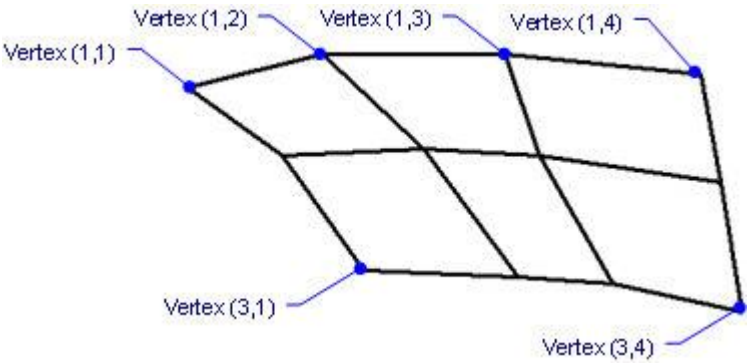
Number of vertices in M direction (between 2 and 256): **(Enter a number.)**

Number of vertices in N direction (between 2 and 256): **(Enter a number.)**

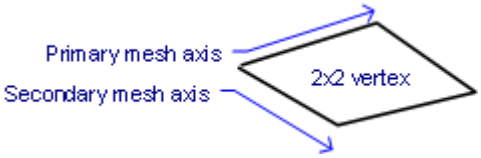
Position of vertex (0, 0): **(Pick a point, or enter x,y,z coordinates.)**

...

Vertex (3, 4): **(Pick another point, or enter x,y,z coordinates.)**



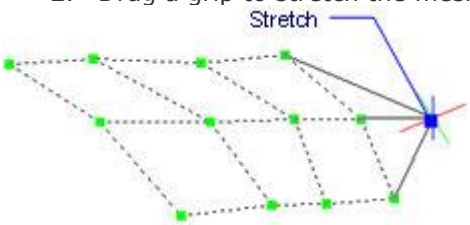
Command Options

Option	Description
Number of vertices in M direction	Specifies the number of vertices in the primary direction. Enter a number between 2 and 256. A single face is made from a 2x2 vertex. 
Number of vertices in N direction	Specifies the number of vertices in the secondary direction. Enter a number between 2 and 256.
Position of vertex (n, n)	Specifies the position of the vertex. Pick a point, or enter x,y,z coordinates.

Grips Editing

3D meshes can be edited directly through grips:

1. Select the 3D mesh . Notice that it has several grips.
2. Drag a grip to stretch the mesh.



Related Commands

3dFace - draws 3D 4-edged faces with optional invisible edges.
PEdit - edits 3D meshes.
PFace - draws 3D multi-sided meshes.

3dOsnap

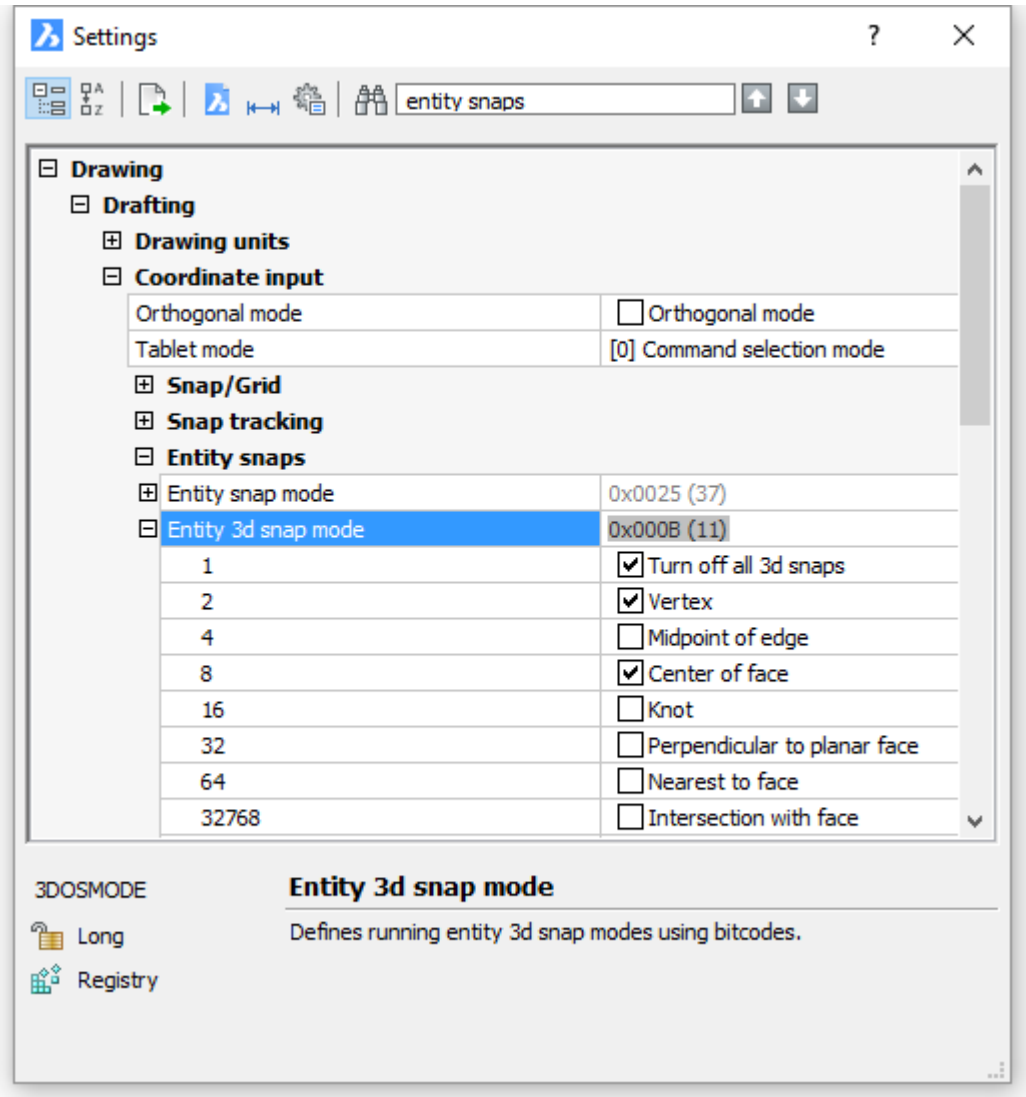
Sets the entity snap modes for 3D entities through the Settings dialog box (short for "3D object snap").

Accessing the Command

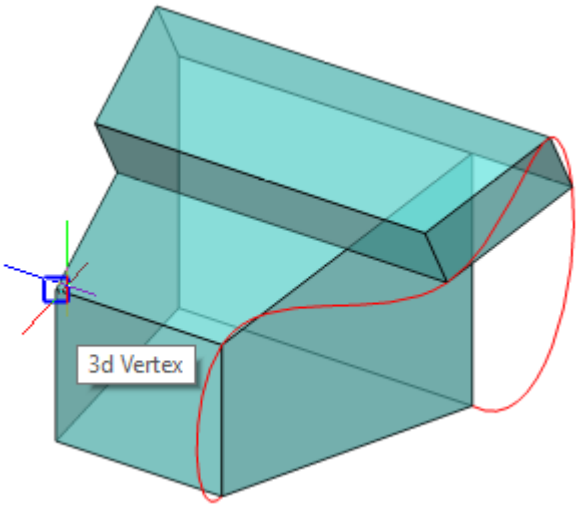
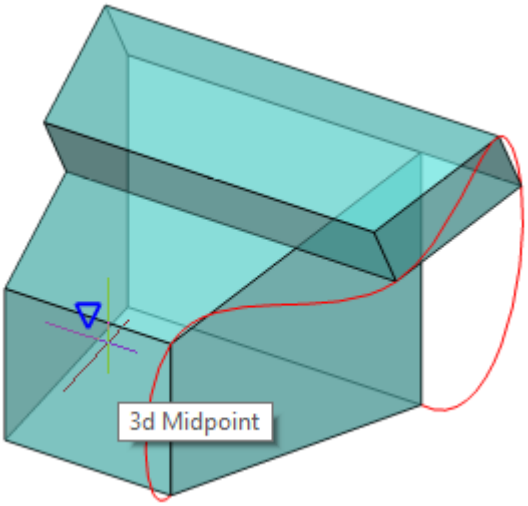
command bar: *3dosnap*

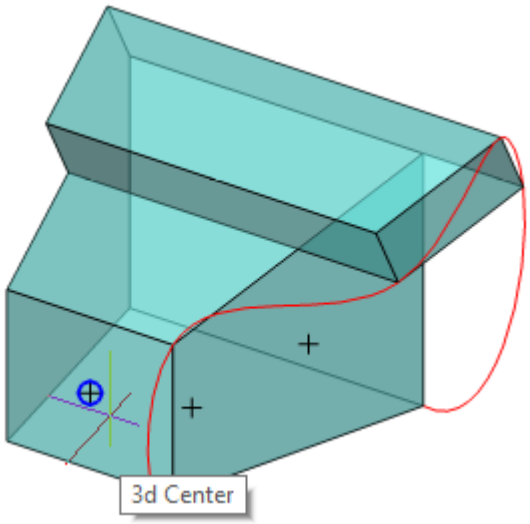
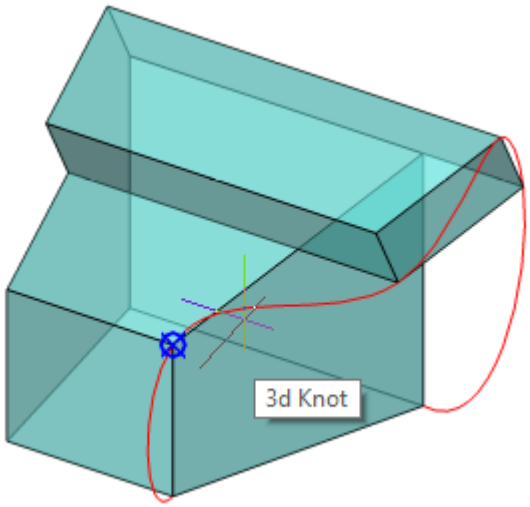
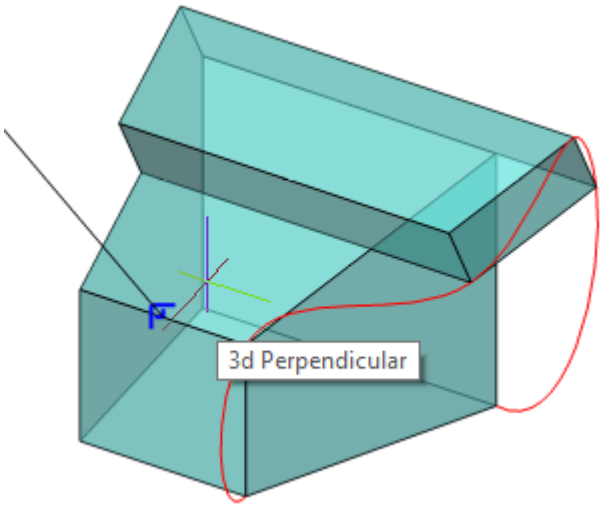
toolbar: *3D Entity Snaps* |

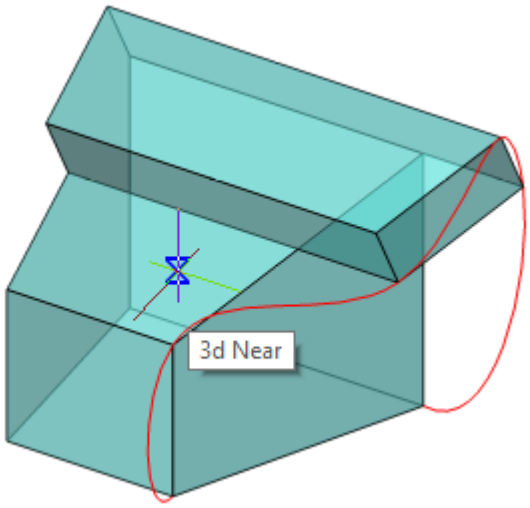
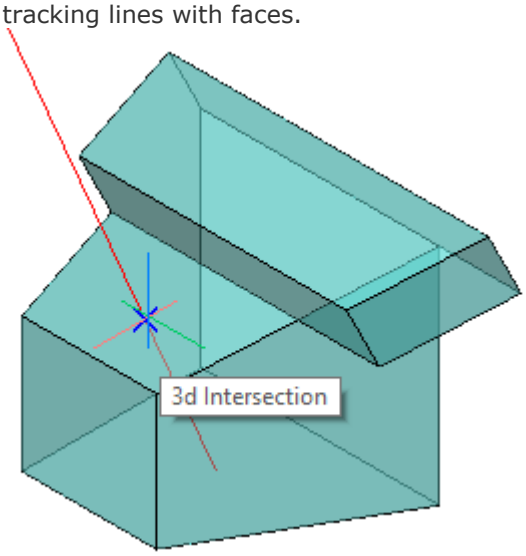
: ***3dOsnap***
Displays a dialog box:



Command Options

Option	Icon	Description
Turn of all 3d snaps		Allows to toggle the currently selected 3d snap modes on/off.
Vertex		Snaps to the closest vertex of a 3d entity. 
Midpoint of edge		Snaps to the midpoint of a face edge. 

Center of face		Snaps to the center of planar or curved 3d faces. 
Knot		Snaps to a knot on a spline. 
Perpendicular to planar face		Snaps to a point perpendicular to a face. 

Nearest to face		Snaps to a point on the face of a 3D entity that is nearest to the cursor. 
3D Intersection		Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. 

Related Commands

-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.

Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

-3dOsnap

Toggles the entity snap modes for 3D entities at the command line (short for "3D object snap").

Accessing the Command

command bar: *-3dosnap*

: ***-3dosnap***
Prompts you in the command bar:
3d Snaps:
ZNEArest/ZVERtex/ZMIDpoint/ZCENter/ZPERpendicular/ZKNOt/ZINTersection/ON/<OFF>: *(Type ON or OFF to toggle the 3d entity snap modes or type one or more options.)*
If you want to set more then one snap mode, separate the names with commas.

Command Options

Option	Description
ZNEArest	Snaps to a point on the face of a 3D entity that is nearest to the cursor.
ZVERtex	Snaps to the closest vertex of a 3d entity.
ZMIDpoint	Snaps to the midpoint of a face edge.
ZCENter	Snaps to the center of 3d face.
ZPERpendicular	Snaps to a point perpendicular to a face.
ZKNOt	Snaps to a knot on a spline.
ZINTersection	Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
ON	Enables the currently selected 3d snap modes.
OFF	Disables the currently selected 3d snap modes.

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

3dPoly

Draws 3D polylines (short for "three dimensional polylines").

Accessing the Command

command bar: *3dpoly*
alias: *3p*
menu bar: *Draw | 3D Polyline*
toolbar: *Draw | Polylines |*

: ***3dpoly***
Prompts you in the command bar:

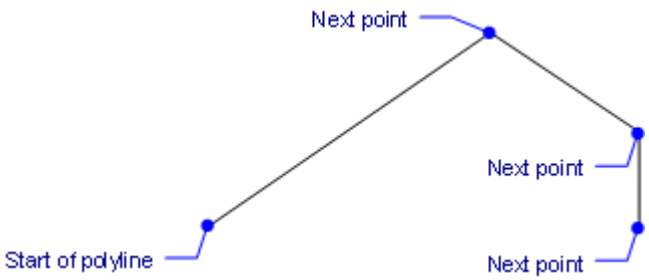
Select start of polyline or [Follow]<ENTER to use last point>: **(Pick a point, or enter x,y,z coordinates, or press Enter to select the last point.)**

Next point: **(Pick another point, or enter x,y,z coordinates.)**

Next point or [Undo]: **(Pick a point, or choose an option.)**

Set next point or [Close/Undo]: **(Pick a point, or choose an option.)**

Set next point or [Close/Undo]: **(Pick a point, or choose an option, or press Enter to stop.)**



You can edit 3D polylines with the **PEdit** command.

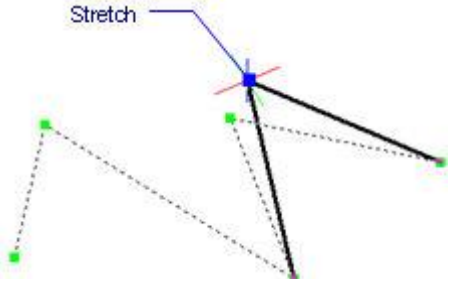
Command Options

Option	Description
Last point	Uses the last point as the start point of the polyline.
Start of polyline	Specifies the starting point of the polyline. Pick a point, or enter x,y,z coordinates.
Follow	Draws the first segment at the same angle as the previously created line or (3D or 2D) polyline segment; move the mouse to indicate the length.
Next point	Specifies the next vertex of the polyline. Pick a point, or enter x,y,z coordinates. Press Enter to exit the command.
Undo	Removes the last segment.
Close	Closes the polyline and ends the command.

Grips Editing

3D polylines can be edited directly through grips:

1. Select the 3D polyline. Notice that it has a grip at the end of each segment.
2. Drag a grip to stretch the segment.



Related Commands

Helix - creates a 2D spiral or a 3D helix.
PLine - draws 2D polylines.
PEdit - edits polylines.
Donut - draws fat circles out of polylines.
Polygon - draws regular polygons out of polylines.
Rectang - draws rectangles out of polylines.

A

About

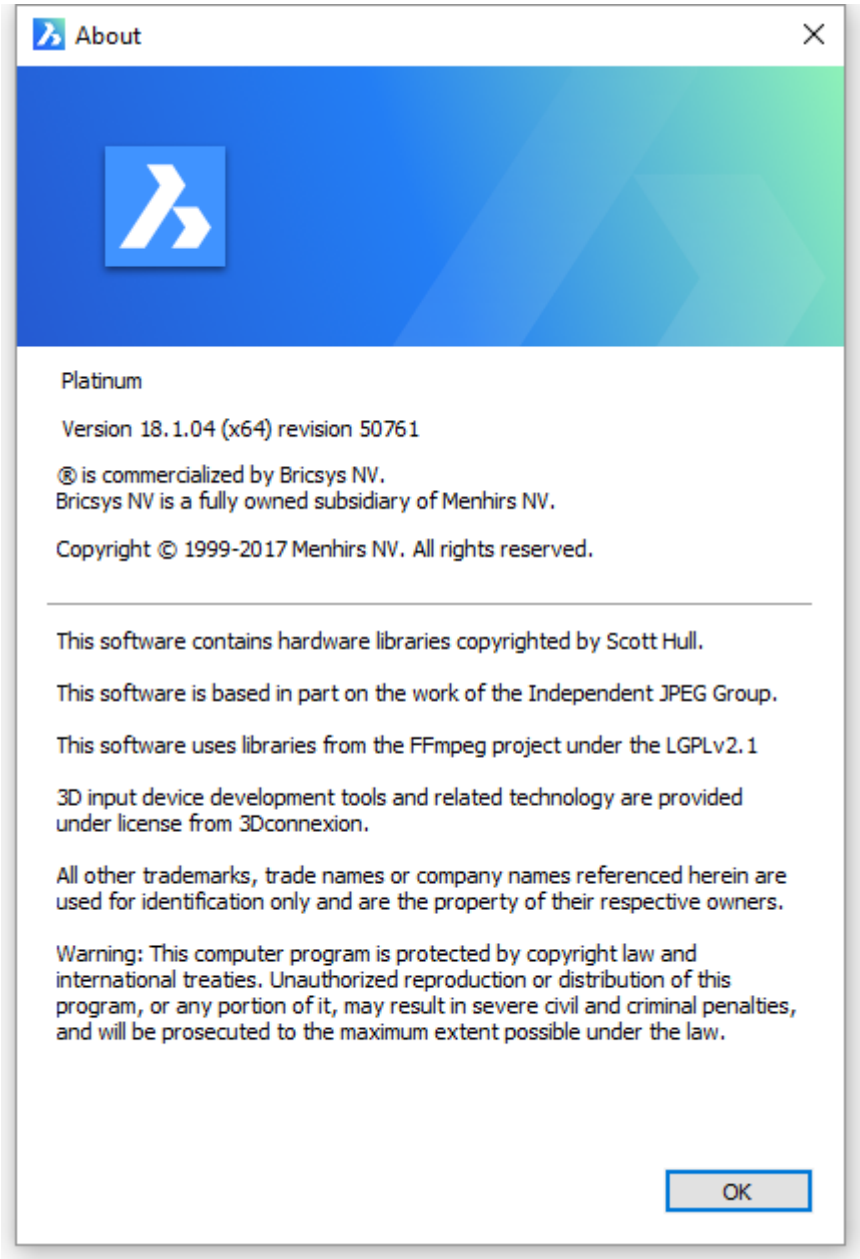
Opens About dialog box, with version and copyright information

Accessing the Command

command bar: *about*
transparent: *'about*
menu bar: *Help | About*

: ***about***

Displays a dialog box:



*Click **OK**.*

Command Options

Option	Description
OK	Dismisses the dialog box.

Related Commands

Status - reports on the state of the drawing.
LicProperties - reports the license number and owner.

AcisIn

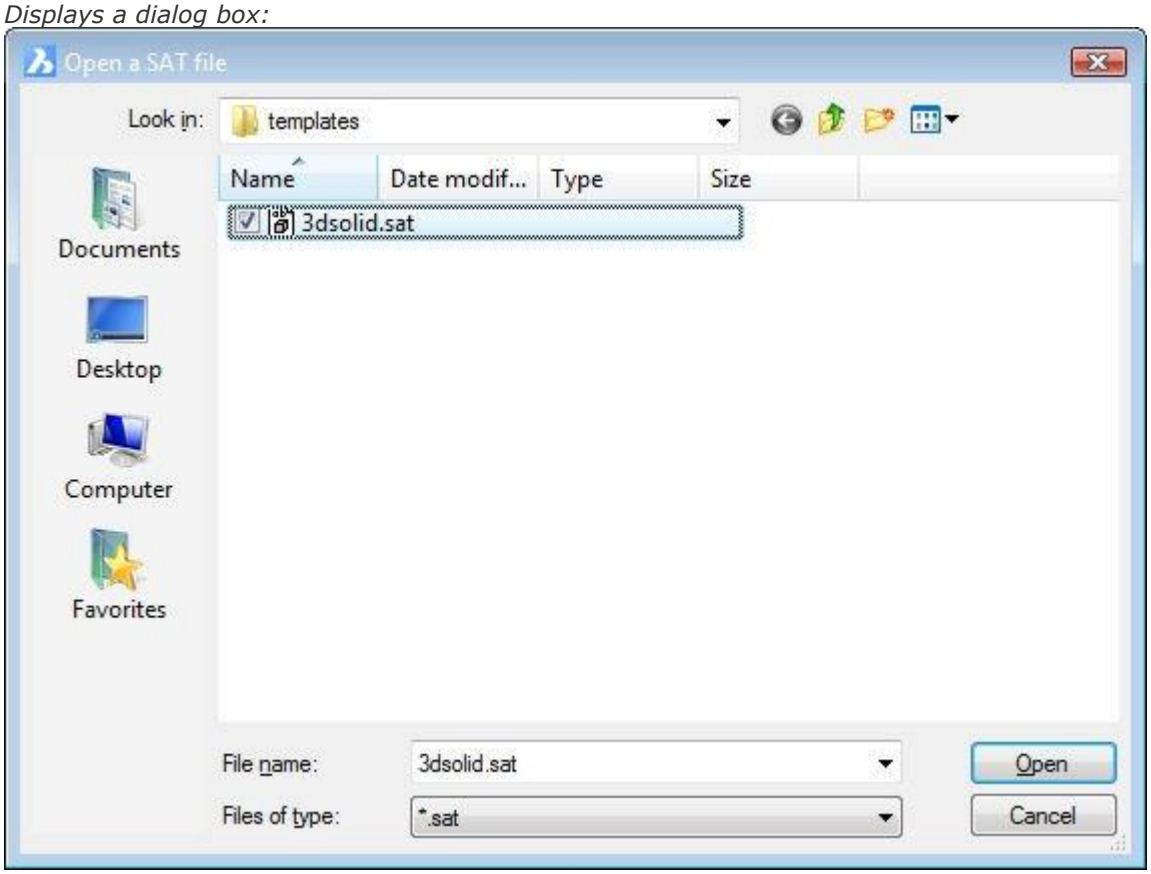
Pro, Platinum Pro, Platinum Pro, Platinum

Imports 3D solids in SAT format (SAT is short for "save as text").

Accessing the Command

command bar: *acisin*
menu bar: *Insert | Acis In*

: ***acisin***
Displays a dialog box:



Choose a SAT file, and then click Open.
The data in the file are converted to 3D solid entities, and then placed on layer 0.

Use this command to import 3D solids from other CAD packages.
Use the **AcisOut** command to create SAT files.

Command Options

Option	Description
File name	Specifies the name of the SAT file. SAT is short for "save as text."
Open	Opens the 3D solid in the current drawing. If you cannot see the entity, use the **Zoom** All command.

Related Commands

3dConvert - converts 3D solid models into surface meshed entities.
AcisOut - exports 3D solids and surface entities in SAT format.

AcisOut

Pro, Platinum Pro, Platinum Pro, Platinum

Exports 3D solids and surface entities in SAT format (SAT is short for "save as text").

Accessing the Command

command bar: *acisout*
menu bar: *Insert | Acis Out*

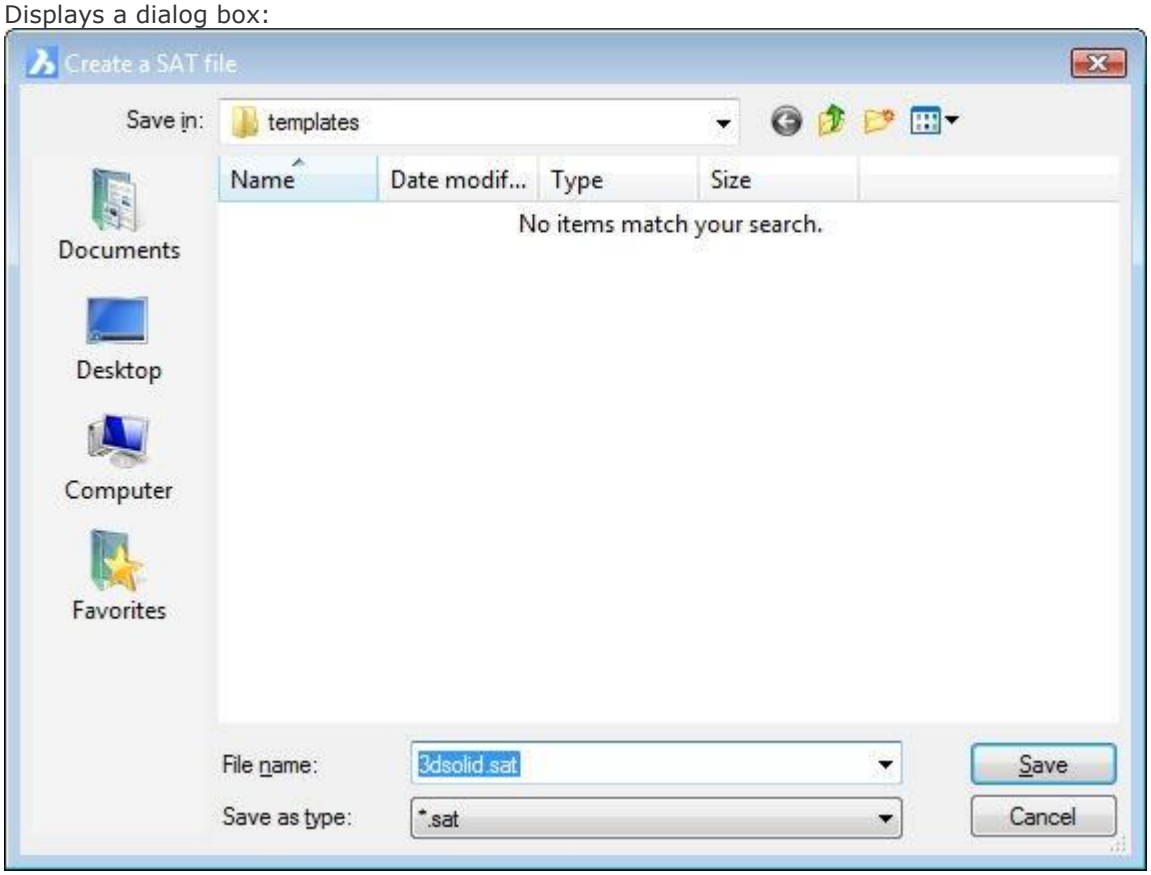
: ***acisout***

Prompts you in the command bar:

Select ACIS entities to save: (Choose one or more 3D solids or surface entities.)

Select ACIS entities to save: (Press Enter to continue the command.)

Displays a dialog box:



Name the SAT file, and then click Save.

The 3D solid entities are converted to ASCII data in the SAT file.

Use this command to export 3D solids to other CAD packages.

Use the **AcisIn** command to import SAT files.

Command Options

Option	Description
File name	Names the SAT file.
Save	Saves the selected 3D solid entities.

Related Commands

3dConvert - converts 3D solid models into surface meshed entities.
AcisIn - imports ACIS solids into the drawing.

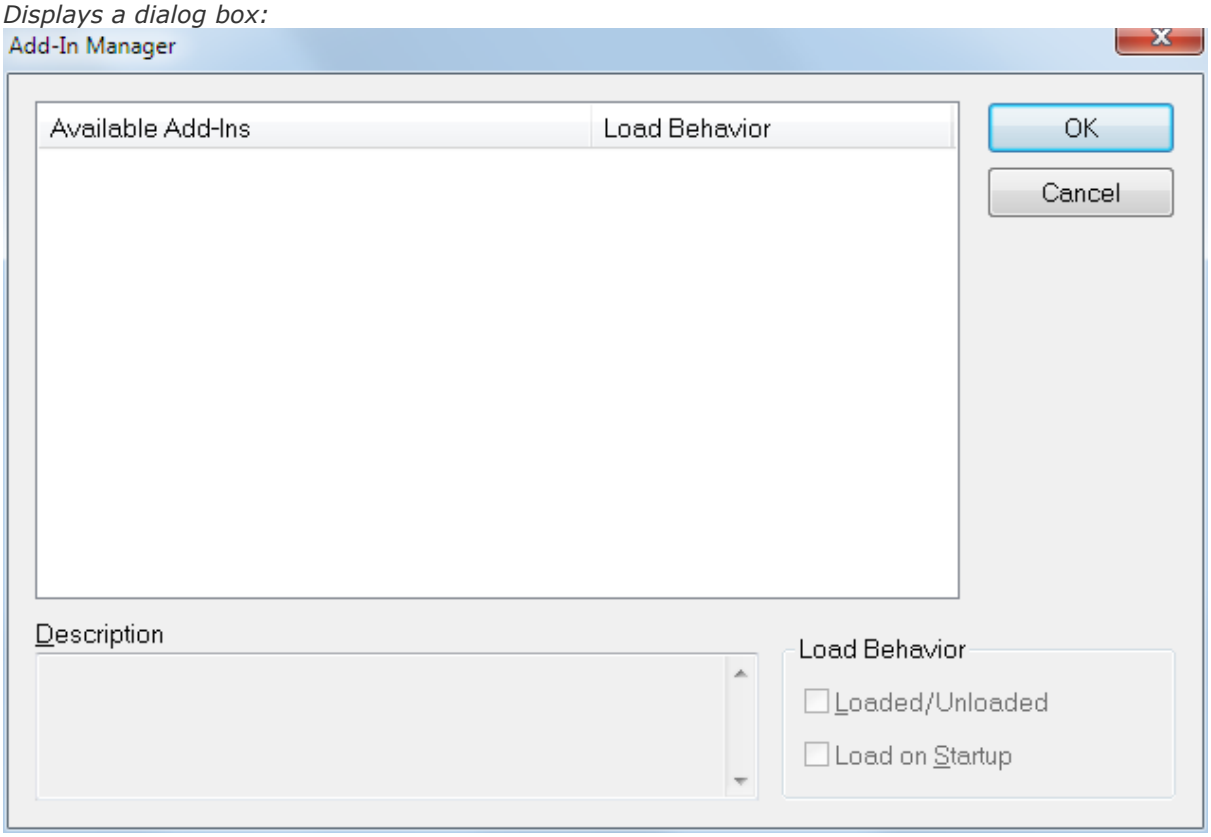
AddInMan

Brings up the VBA COM Add-In Manager.

Accessing the Command

command bar: *addinman*
shortcut: *F11*
alias: *Alias*
menu bar: *Tools | VBA | Add-in Manager*

: ***addinman***
Displays a dialog box:



*Change the load behavior of add-ins, and then click **OK**.*

Command Options

Option	Description
Available Add-Ins	Lists the names of VBA COM modules available to BricsCAD. VBA COM is short for Visual Basic for Applications Common Object Model.
Load Behavior	Determines how modules are loaded: • **Loaded/Unloaded** toggles the module between loaded and unloaded. • **Load on Startup** loads modules at the same time as BricsCAD starts.
OK	Saves the changes, and then dismisses the dialog box.

Cancel	Undoes the changes, and then dismisses the dialog box.

Related Commands

AppLoad - selects external routines to load when BricsCAD starts.
Commands - lists the names of internal and external commands.
CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

AddSelected

Creates a new entity of the same type as an existing entity.

Accessing the Command

command bar: *addselected*
toolbar: *Standard* |

: ***addselected***
Prompts you in the command bar:

Select an entity: ***(Select an entity in the drawing.)***
Launches the command to create the selected entity type.

Note: Some entities might not yet be supported.

Command Options

No options

Related Commands

All entity creation commands.

Ai_Box

Draws 3D boxes as mesh surfaces.

Accessing the Command

command bar: *ai_box*
menu bar: *Model | Meshes | Box*
toolbar: *Meshes |*

: ***ai_box***
Prompts you in the command bar:

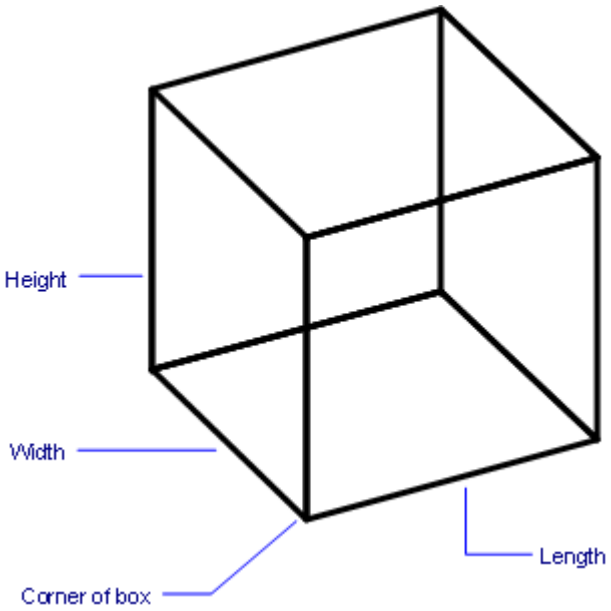
Corner of box <0,0,0>: (Pick a point, or enter coordinates.)

Length of side of box: (Pick a point, or enter a length.)

Set width of box or [Cube]: (Pick a point, enter a width, or type C.)

Height of box: (Enter a height.)

Rotation angle for box <0>: (Enter a rotation angle, or press Enter for none.)



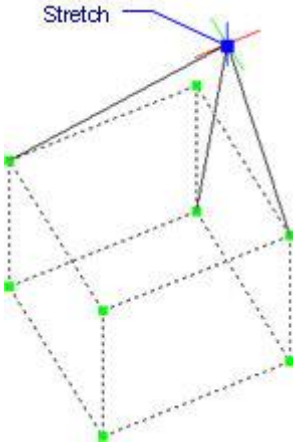
Command Options

Option	Description
Corner of box	Specifies the lower-left corner of the box.
Length of side of box	Specifies the length of one side, along the positive x-direction.
Width of box	Specifies the width of one side, along the positive y-direction.
Height of box	Specifies the height of one side, along the positive z-direction.
Rotation angle for box	Specifies the counterclockwise rotation angle about the corner (initial pick point).
Cube	Draws a box with all sides of equal length.

Grips Editing

Mesh-formed boxes can be edited directly through grips:

1. Select the box. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

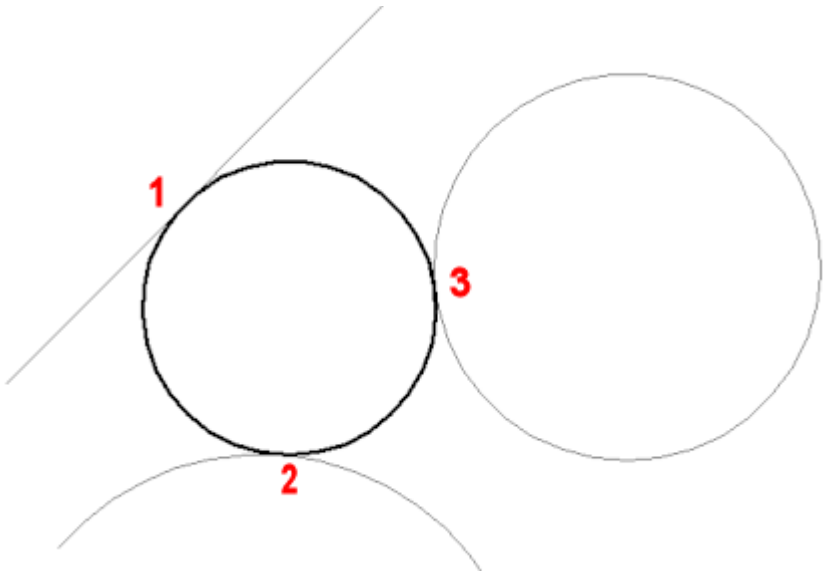
Ai_CircTan

Draws a circle tangent to three entities.

Accessing the Command

command bar: *ai_circtan*

: ***ai_circtan***
Prompts you in the command bar:
Enter Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)**
Enter second Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)**
Enter third Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)**



Command Options

No command options.

Related Commands

Circle - draws circles by several methods.

Ai_Cone

Draws 3D cones as mesh surfaces, with round or elliptical bases

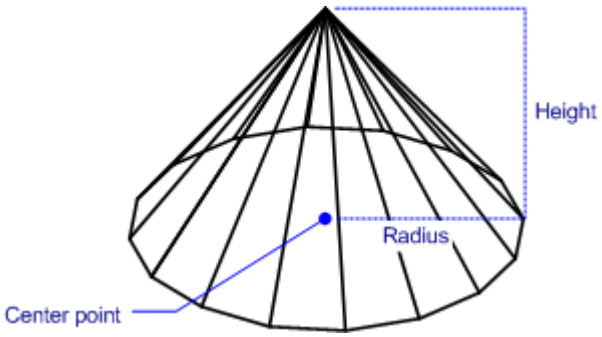
Accessing the Command

command bar: *ai_cone*
menu bar: *Model | Meshes | Cone*
toolbar: *Meshes |*

: ***ai_cone***

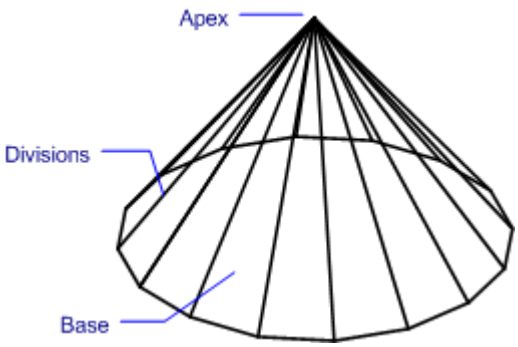
Prompts you in the command bar:

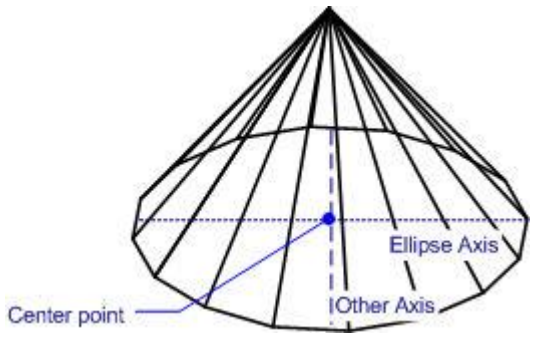
Elliptical/Divisions/<Center of base of cone> <0,0,0>: (Pick a point, enter coordinates, or enter an option.)
Diameter/<**Radius of base**>: (Pick a point, enter coordinates, or type D.)
Apex/<**Height of cone**>: (Pick a point, enter coordinates, or type A.)



Command Options

Option	Description
Center of base of cone	Specifies the center point of the cone's base.
Radius of base	Specifies the radius of the cone's base.
Height of cone	Specifies the height of the cone (in the z direction); enter a negative number to draw the cone upside down.

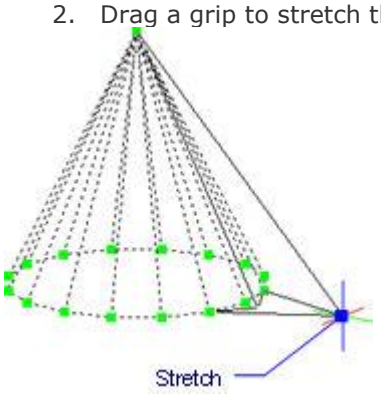


Option	Description
Apex	Specifies coordinates for the top of the cone, an alternative to specifying the height. Prompts you: Apex of cone - specify the top of the cone by pointing or entering a height.
Diameter	Specifies the diameter of the cone's base: Diameter of base - specify the diameter by pointing or entering a value.
Divisions	Changes the number of isolines used to simulate the cone's curved surface. Prompts you: Number of divisions - change the number of isolines from the default of 1.
Elliptical  	Specifies that the cone should have an elliptical base. Prompts you: Center **specifies the center of the ellipse.** First end of ellipse axis - **specify the endpoint of one ellipse axis.** Second end of ellipse axis - **specify the other endpoint of the first axis.** Other axis - **specify the length of the other (second) axis.**

Grips Editing

Mesh-formed cones can be edited directly through grips:

1. Select the cone. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_Cylinder

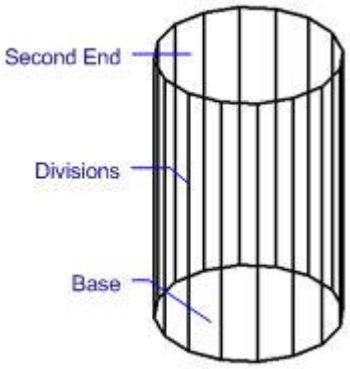
Draws 3D cylinders as 3D mesh surfaces, with round or elliptical bases.

Accessing the Command

command bar: *ai_cylinder*
menu bar: *Model | Meshes | Cylinder*
toolbar: *Meshes |*

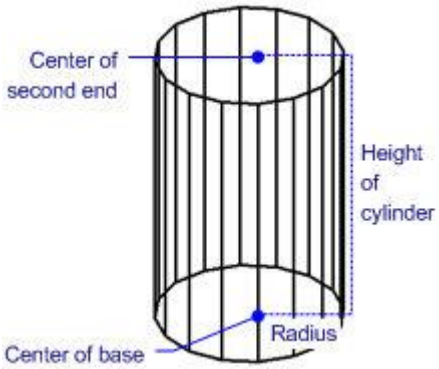
: ***ai_cylinder***

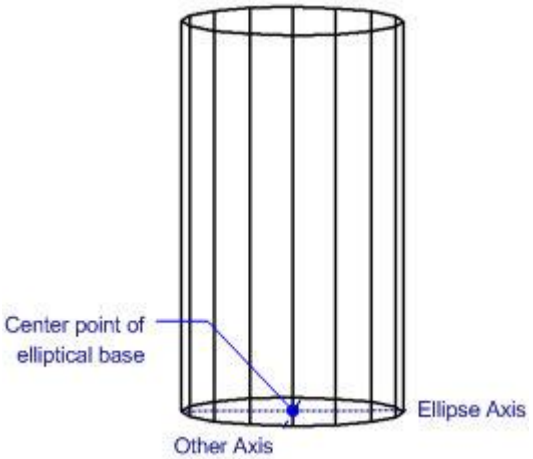
Prompts you in the command bar:
Elliptical/**Divisions**/<**Center of base of cylinder**> <0,0,0>: (Pick a point, enter coordinates, or enter an option.)
Diameter/<**Radius of cylinder**>: (Specify radius, or enter D.)
Center of second end/<**Height of cylinder**>: (Specify height, or enter C.)



Command Options

Option	Description
Center of base of cylinder	Specifies the center point of the cylinder's base; the base is drawn on the current x,y-plane.
Radius of cylinder	Specifies the radius of the cylinder.
Height of cylinder	Specifies the height of the cylinder (in the z direction); enter a negative number to draw the cylinder downwards.

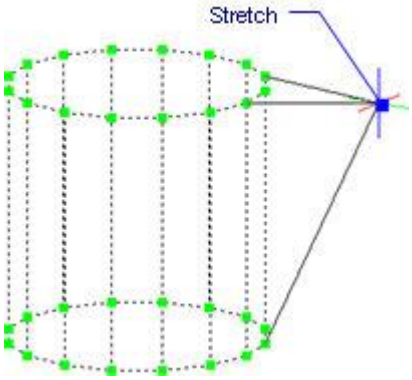


Option	Description
Center of second end	Specifies coordinates for the other of the cylinder, an alternative to specifying the height. Prompts you: Center of second end -s specify the other end of the cylinder by pointing or entering a height.
Diameter	Specifies the diameter of the cylinder's base. Prompts you: Diameter of cylinder - specify the diameter by pointing or entering a value.
Divisions	Changes the number of isolines used to simulate the cylinder's curved surface. Prompts you: Number of divisions - change the number of isolines from the default of 16.
Elliptical  	**Specifies that the cylinder should have an elliptical cross-section. Prompts you:** Center **specifies the center of the ellipse.** First end of ellipse axis - **specify the endpoint of one ellipse axis.** Second end of ellipse axis - **specify the other end point of the first axis.** Other axis - **specify the length of the second axis.**

Grips Editing

Mesh-formed cylinders can be edited directly through grips:

1. Select the cylinder. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_Dish

Draws the bottom half of a sphere as a 3D mesh surface.

Accessing the Command

command bar: *ai_dish*
menu bar: *Model | Meshes | Dish*
toolbar: *Meshes |*

: ***ai_dish***

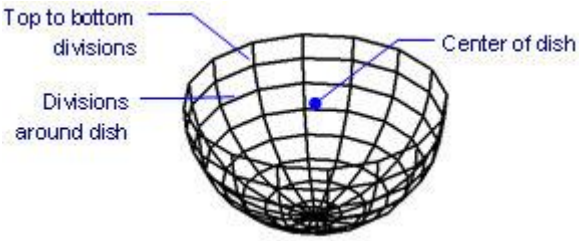
Prompts you in the command bar:

Center of dish: (Pick a point or enter coordinates.)

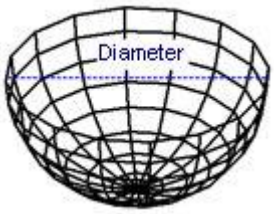
Set radius of dish or [**Diameter**]: (Pick a point, enter coordinates, or enter a value.)

Number of divisions around dish <16>: (Enter a number, or press Enter to keep the default value.)

Number of top-to-bottom divisions <8>: (Enter a number, or press Enter to keep the default value.)



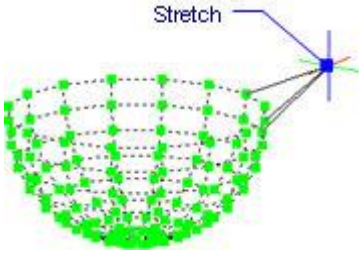
Command Options

Option	Description
Center of dish	Specifies the center point of the dish's open end; the end is drawn on the current x,y-plane.
Radius of dish	Specifies the radius of the dish.
Number of divisions around dish	Changes the number of horizontal isolines used to simulate the dish's curved surface.
Number of top-to-bottom divisions	Changes the number of vertical isolines from the default of 8.
Diameter 	Specifies the diameter of the dish. Prompts you: Diameter of dish - specify the diameter by pointing or entering a value.

Grips Editing

Mesh-formed dishes can be edited directly through grips:

1. Select the dish. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_Dome

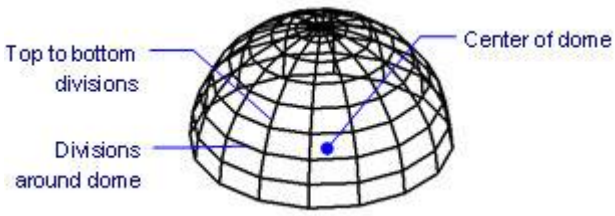
Draws the upper half of a sphere as a 3D mesh surface.

Accessing the Command

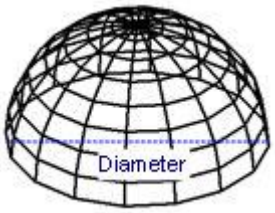
command bar: *ai_dome*
menu bar: *Model | Meshes | Dome*
toolbar: *Meshes |*

: ***ai_dome***

Prompts you in the command bar:
Center of dome: (Pick a point or enter coordinates.)
Set radius of dome or [**Diameter**]: (Pick a point, enter coordinates, or type a value.)
Number of divisions around dome <16>: (Enter a number, or press Enter to keep the default value.)
Number of top-to-bottom divisions <8>: (Enter a number, or press Enter to keep the default value.)



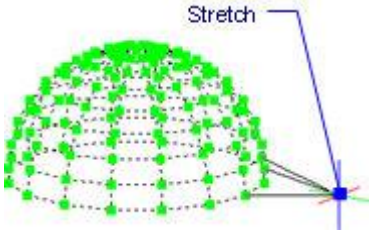
Command Options

Option	Description
Center of dome	Specifies the center point of the dome's open end; the end is drawn on the current x,y-plane.
Radius of dome	Specifies the radius of the dome.
Number of divisions around dome	Changes the number of horizontal isolines used to simulate the dome's curved surface.
Number of top-to-bottom divisions	Changes the number of vertical isolines from the default of 8.
Diameter 	Specifies the diameter of the dome. Prompts you: Diameter of dome - specify the diameter by pointing or entering a value.

Grips Editing

Mesh-formed domes can be edited directly through grips:

1. Select the dome. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_EdgeSurf

This command is obsolete. It still exists for backward compatibility only.
Please use Edgesurf instead.

Ai_Pyramid

Draws a pyramid or tetrahedron as a 3D mesh surface.

Accessing the Command

command bar: *ai_pyramid*
menu bar: *Model | Meshes | Pyramid*
toolbar: *Meshes |*

: ***ai_pyramid***

Prompts you in the command bar:

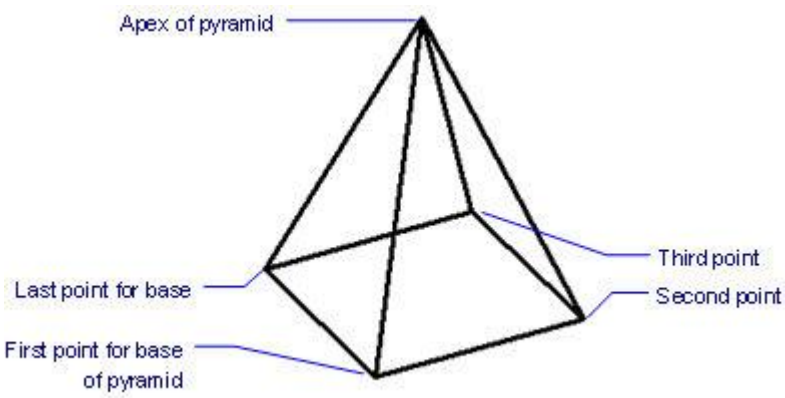
First point for base of pyramid: (Pick a point, or enter coordinates.)

Second point: (Pick another point, or enter coordinates.)

Third point: (Pick a third point, or enter coordinates.)

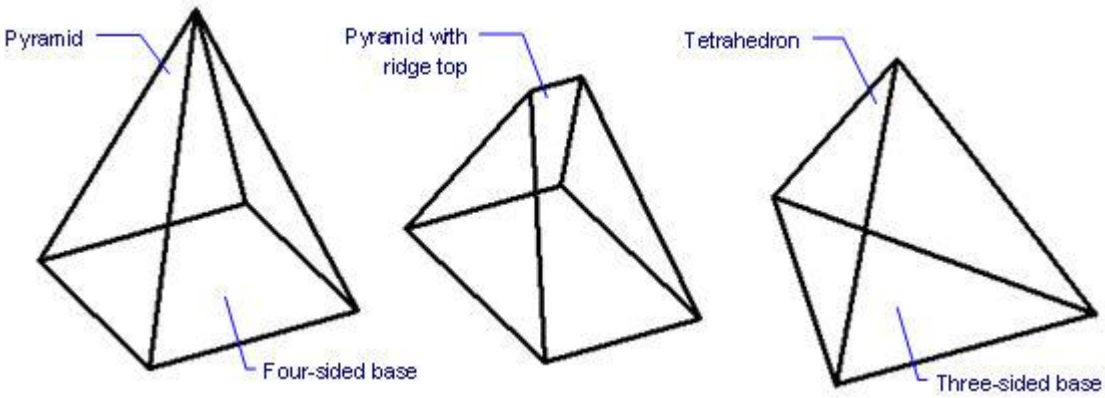
Set last point for base or [**Tetrahedron**]: (Pick the last point, enter coordinates, or type T.)

Set apex of pyramid or [**Ridge**/**Top**]: (Pick a point, or enter coordinates, or enter an option.)



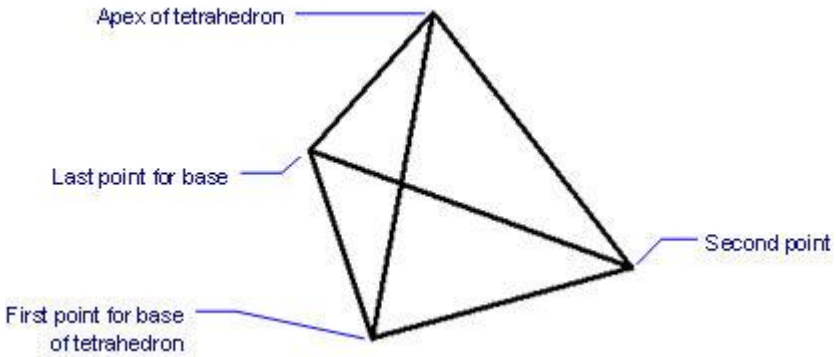
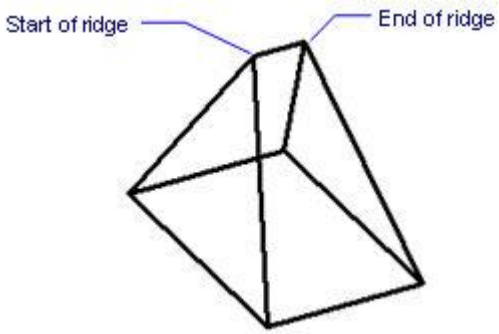
This command draws:

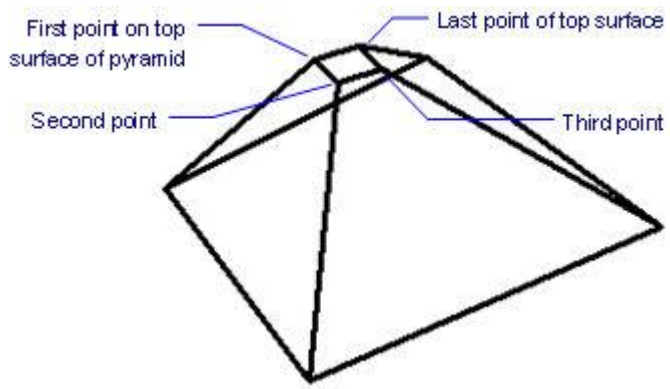
- Pyramids with four-sided base and four sides; the base does not need to be an equilateral.
- Tetrahedrons with triangular base and three sides; the base can be scalene.
- Pyramids with pointed, ridged, or flat tops.



To draw a rotated pyramid, specify the angle during the Apex option.

Command Options

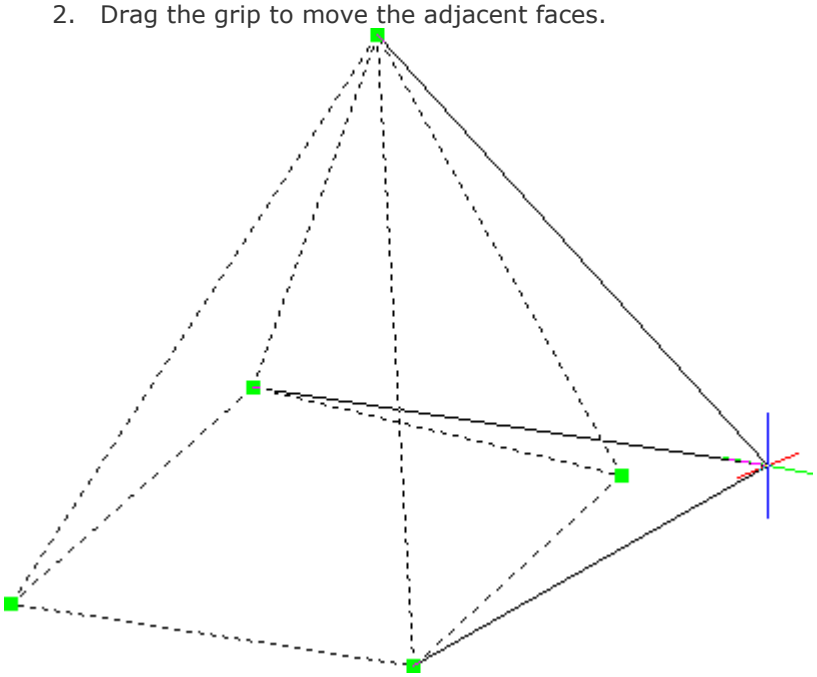
Option	Description
First point for base of pyramid	Specifies the first corner of the pyramid's base.
Second point	Specifies the second corner of the base.
Third point	Specifies the third point of the pyramid's base, or the last point for a tetrahedron's base.
Last point for base	Specifies the fourth point for the pyramid's base.
Apex of pyramid	Specifies the height and angle of the pyramid; draws a pointy top.
Tetrahedron 	Draws tetrahedrons; prompts you: Apex of tetrahedron - specify the top of the tetrahedron.
Ridge 	Draws a ridge at the top of the pyramid. Prompts you: Start of ridge specify the height and location of one end of the ridge. End of ridge specify the height and location of the other end of the ridge.
Top surface	Draws a flat surface at the top of the pyramid; prompts you: First point on top surface of pyramid - specify the starting corner of the pyramid's flat top.

	Second point - specify the next corner. Third point - specify the third corner for pyramid, and the last corner for a tetrahedron. Last point of top surface - specify the fourth corner for the pyramid.

Grips Editing

Mesh-formed pyramids can be edited directly through grips:

1. Select the pyramid. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_RevSurf

This command is obsolete. It still exists for backward compatibility only.
Please use RevSurf instead.

Ai_RuleSurf

This command is obsolete. It still exists for backward compatibility only.
Please use RuleSurf instead.

Ai_Sphere

Draws a sphere as a 3D mesh surface.

Accessing the Command

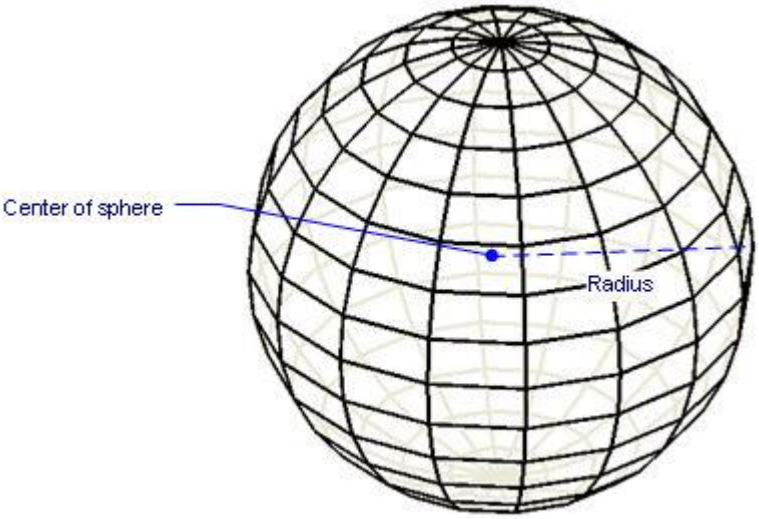
command bar: *ai_sphere*
menu bar: *Model | Meshes | Sphere*
toolbar: *Meshes |*

: ***ai_sphere***
Prompts you in the command bar:

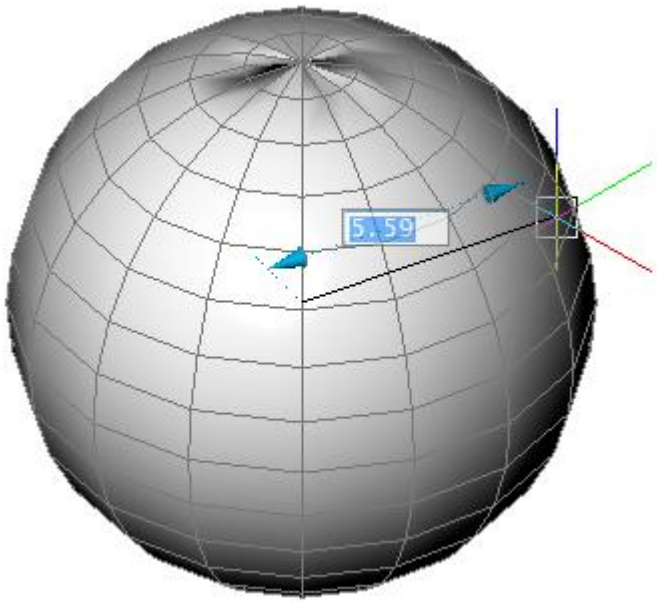
Center of sphere: (Pick a point, or enter coordinates.)
Set radius of sphere or [Diameter]: *(Specify a distance, or type D.)*

Number of longitudinal segments <16>: (Type a value or press Enter to accept the default number.)

Number of latitudinal segments <16>: (Type a value or press Enter to accept the default number.)



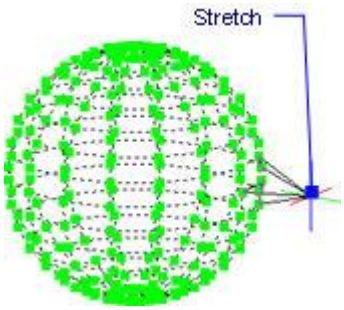
If Dynamic Dimensions are active, you can enter the radius in the dynamic dimension entry field.



Command Options

Option	Description
Center of sphere	Specifies the center point of the sphere; enter x,y- or x,y,z-coordinates, or pick a point in the drawing.
Radius of sphere	Specifies the radius.
Diameter	Specifies the diameter.
Number of longitudinal segments	Specifies the number of segments of the circles perpendicular to the vertical axis of the sphere.
Number of latitudinal segments	Specifies the number of segments of the circles through to the vertical axis of the sphere.

Grips Editing



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_TabSurf

This command is obsolete. It still exists for backward compatibility only.
Please use TabSurf instead.

Ai_Torus

Draws a torus as a 3D mesh surface.

Accessing the Command

command bar: *ai_torus*
menu bar: *Model | Meshes | Torus*
toolbar: *Meshes |*

: ***ai_torus***
Prompts you in the command bar:

Center of whole torus: <0,0,0> (Pick a point, or enter coordinates.)

Set Radius of whole torus or [**Diameter**]: (Specify the radius, or type D.)

Set radius of body of torus [**Diameter**]: (Specify the radius, or type D.)

Segments around body of torus <16>: (Type a value or press Enter to accept the default number.)

Segments around whole torus <16>: (Type a value or press Enter to accept the default number.)

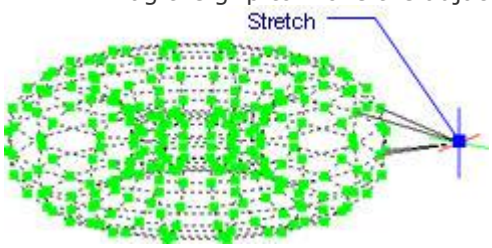
Command Options

Option	Description
Center of whole torus	Specifies the center of the torus; you can press Enter to accept the default coordinates.
Radius of whole torus	Specifies the overall radius of the torus; note that the distance is measured to the center of the tube (body).
Radius of body of torus	Specifies the radius of the torus body, or tube. It is not necessary for the body radius to be smaller than the whole radius.
Diameter	Specifies the diameters of the whole torus and/or body.
Segments around body of torus	Specifies the number of segments of the circles around the body of torus.
Segments around body whole torus	Specifies the number of segments of the circles around the whole torus.

Grips Editing

Mesh-formed torus surfaces can be edited directly through grips:

1. Select the torus. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

Ai_Wedge

Draws a wedge as a 3D mesh surface.

Accessing the Command

command bar: *ai_wedge*
menu bar: *Model | Meshes | Wedge*
toolbar: *Meshes |*

: ***ai_wedge***

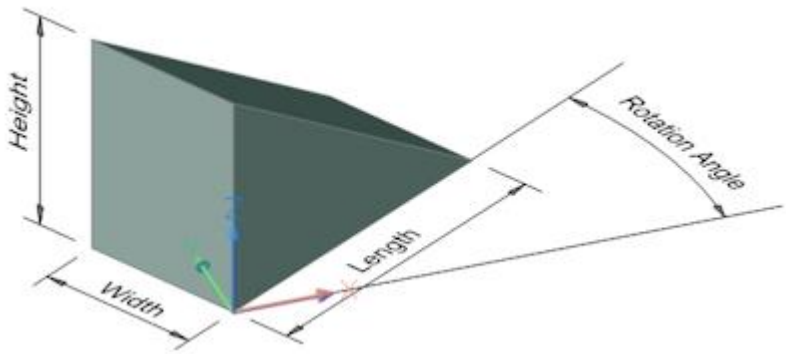
Prompts you in the command bar:

Corner of wedge: (Pick a point, or enter coordinates.)
Length of wedge: (Pick a point, or enter a value.)

Width of wedge: (Pick a point, or enter a value.)

Height of wedge: (Pick a point, or enter a value.)

Rotation angle for wedge <0>: (Pick a point, or enter a value.)



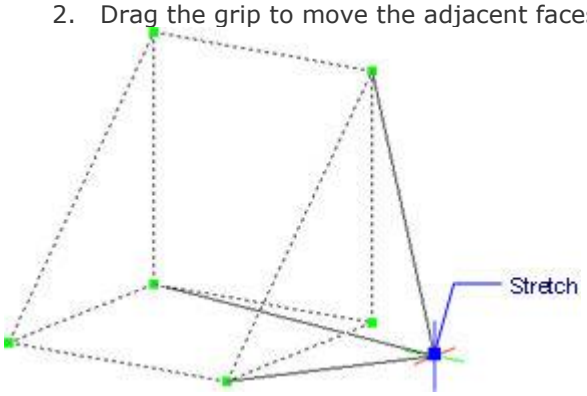
Command Options

No options.

Grips Editing

Mesh-formed wedges can be edited directly through grips:

1. Select the wedge. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
Ai_TabSurf and TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.

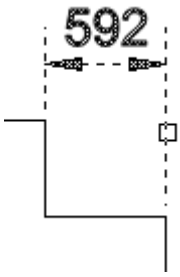
AiDimFlipArrow

Flips individual arrows of a dimension.

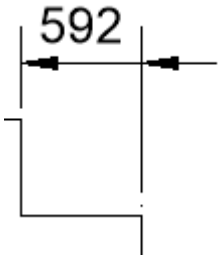
Accessing the Command

command bar: *aidimpfliparrow*
menu: *Dimension | Flip dimension arrowhead*
toolbar: *Dimensions |*

: *aidimpfliparrow*
Prompts you in the command bar:
Select entities: ***(Click the extension line of the dimension arrow you want to flip.)***



The arrow flips to the other side of the extension line.



Command Options

No options.

Related Commands

AiDimPrec - Changes the displayed precision of existing dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimAngular - places angular dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.

AiDimPrec

Changes the displayed precision of existing dimensions.

Accessing the Command

command bar: *aidimprec*

: ***aidimprec***
Prompts you in the command bar:

Enter option [0/1/2/3/4/5/6] <0>: **(Choose an option or press Enter to accept the default value.)**

Select entities: **(Select a dimension entity. Dimension entities will be accepted only; other entity types are ignored.)**
Entities in set: 1

Select entities: **(Select more dimension entities or press Enter.)**

Command Options

Option	Description
0/1/2/3/4/5/6	Sets the number of decimal places.

Related Commands

AiDimFlipArrow - flips individual arrows of a dimension.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimAngular - places angular dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.

AiDimStyle

Creates Dimension Styles based on the properties of selected Dimension, Tolerance or Leader entities.

Accessing the Command

command bar: *aidimstyle*

: ***aidimstyle***
Prompts you in the command bar:

Enter option [Apply/Save] <Save>: **(Choose an option.)**

Command Options

Option	Description
Save	Creates a new dimension style based on the properties of selected Dimension, Tolerance or Leader entities. Prompts you: **Select entities: (Select a Dimension, Tolerance or Leader entity.)** Style name to save to: **(Type a name for the new dimension style.)**
Apply	Applies a dimension style to selected entities. Prompts you: Style name to apply - type a dimension style name. Select entities - select a Dimension, Tolerance or Leader entity. Entities in set: 1 Select entities - select a Dimension, Tolerance or Leader entity or press Enter.

Related Commands

-DimStyle - creates and modifies dimensions styles at the command line.
DimStyle - creates and modifies dimension styles, through the Drawing Explorer.

Align

Aligns one entity with another through moving, rotating, and scaling.

Accessing the Command

command bar: *align*
alias: *al*
menu bar: *Modify | Align*
toolbar: *Modify |*

: ***align***

Prompts you in the command bar:

Select entities: (Select one or more source entities; do not select destination entities.)

Entities in set: 1

Select entities: (Press Enter to end entity selection.)

Specify first source point: (Pick a point on or near the source entity.)

Specify first destination point: (Pick the matching point on or near the destination entity.)

Specify second source point: (Pick another point on or near the source entity.)

Specify second destination point: (Pick the matching point on or near the destination entity.)

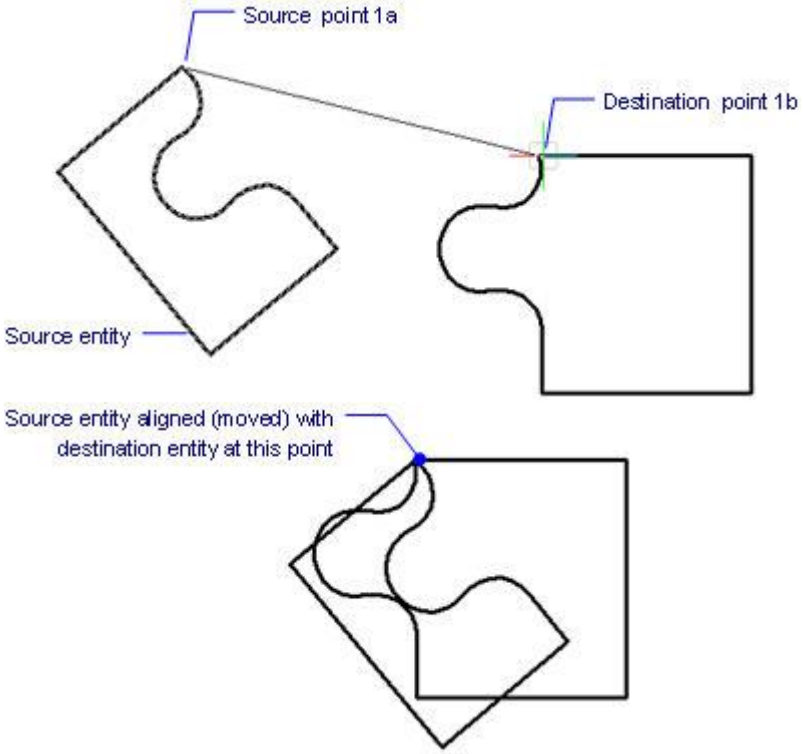
Specify third source point: (Press Enter for 2D entities; pick source and destination points for 3D alignment.)

Scale objects based on alignment points [Yes/No] <No>: (Enter Y or N.)

Command Options

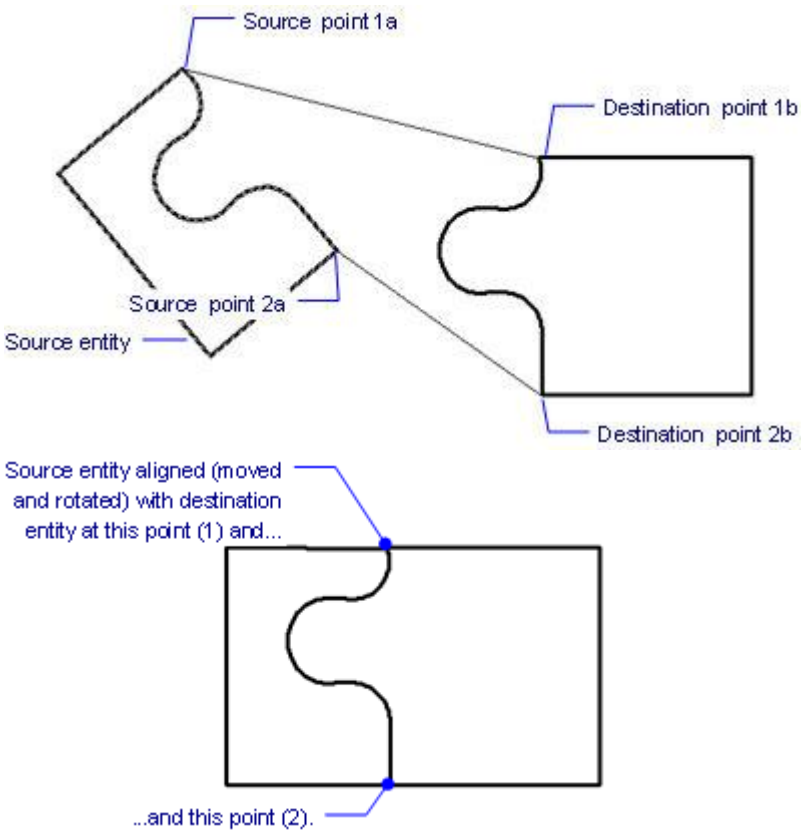
Option	Description
Select entities	Specifies the entities that are to be aligned; do not select destination entities; otherwise they are also move.

To move entities, specify one pair of source and destination points to move entities:



Option	Description
Specify first source point	Specifies the source alignment point, such as #1a in the figure above.
Specify first destination point	Specifies the destination alignment point, such as #1b in the figure.

To move and rotate entities, specify a second pair of source and destination points to move and rotate entities:



Option	Description
Specify second source point	Specifies the second source alignment point, such as #2a in the figure above. If you press Enter at this prompt, the source object moves.
Specify second destination point	Specifies the matching destination point, such as #2b in the figure.

To move and rotate 3D entities, specify a third pair of source and destination points to move and rotate 3D entities.

Option	Description
Specify third source point	Specifies the source point for 3D alignment; press Enter to move and rotate 2D entities.
Specify third destination point	Specifies the matching destination point.

To scale the source entities:

Option	Description
Scale objects based on alignment points	**Optionally scales the source object:** • Yes - scales objects relative to the picked points. • No - does not scale objects.

Procedures

Aligning an entity in 2D
Aligning an entity in 3D

Related Commands

Move - moves entities in 3D.
Rotate - rotates entities in 2D.
Rotate3D - rotates entities in 3D.
Scale - scales entities.

AlignSpace

Adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.
This command applies to paper space only (TILEMODE is set to 0).

Accessing the Command

command bar: *alignspace*

: ***alignspace***
Prompts you in the command bar:

First alignment point in Model space: **(If the current layout consists of a single viewport, specify a point in the viewport; else, click inside the viewport you want to align, then specify the first alignment point.)**

Second alignment point in Model space or <None>: **(Specify a second point in the same viewport.)**

First alignment point in Paper space: **(Specify a point in paper space - this point will be made coincident with the first alignment point.)**

Second alignment point in Paper space: **(Specify a point in paper space - this point will be mad coincident with the second alignment point.)**

Activate the viewport which will be used for alignment. **(Press Enter to accept the current viewport, or click in another viewport, then press Enter.)**

Command Options

Option	Description
None	Press Enter to skip the selection of the second alignment point in Mode space. The selected point will be made coincident with the alignment point in Paper space. You will not be prompted to select a second alignment point in Paper space.

Procedures

To align model space and paper space points.

Related Commands

MvSetup - creates multiple paper space viewports; aligns, rotates and scales paper space viewports.
MView - creates one or more viewports in layout tabs.
MSpace - switches to model space.
PSpace - switches to paper space.

AniPath

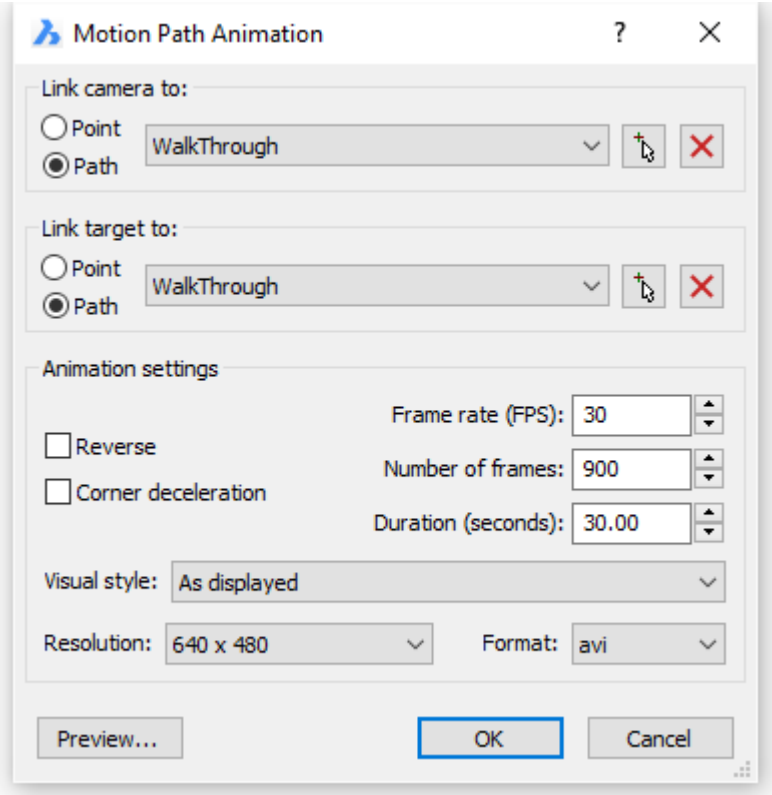
Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

Accessing the Command

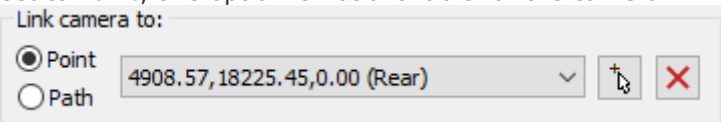
command bar: *anipath*
menu bar: *View | Motion Path Animations*
ribbon: *View | Motion Path Animations |*

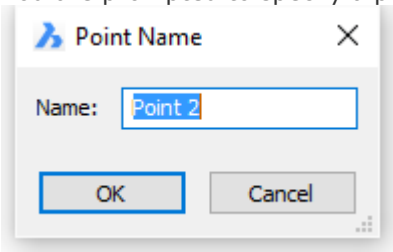
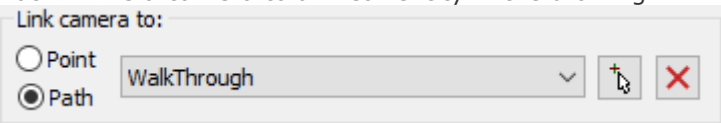
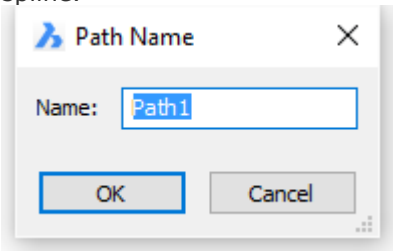
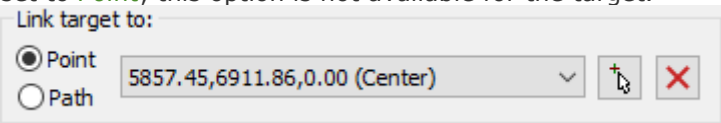
: ***anipath***

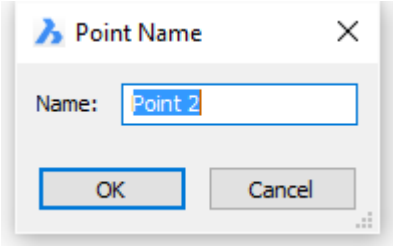
Displays a dialog box:

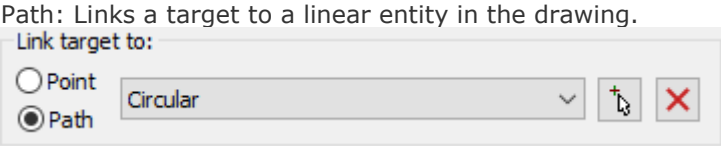


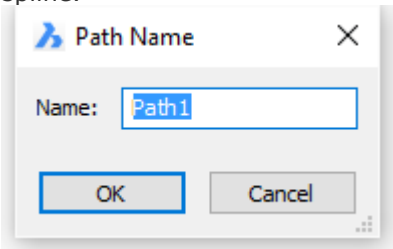
Command Options

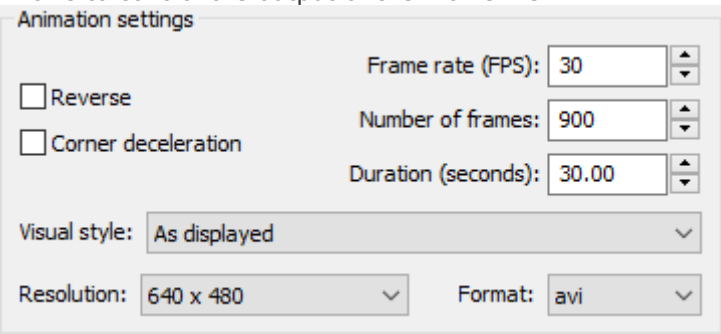
Option	Description
Link camera to	Defines the location of the camera. Select an option: Point: Links a camera to a static point in the drawing. If *Link target to* is set to *Point*, this option is not available for the camera.  Do one of the following: • Click the list button, and choose a previously saved point.

	• Click the *Pick* ( ) button. You are prompted to specify a point in the drawing.  Type a name for the point, then click the *OK* button. The point is added to the points list. • Click the *Delete* button ( ) to remove the currently selected point from the points list. Path: Links a camera to a linear entity in the drawing.  Do one of the following: • Click the list button, and choose a previously saved path. • Click the *Pick* ( ) button. You are prompted to pick a linear entity in the drawing. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.  Type a name for the path, then click the *OK* button. The path is added to the paths list. • Click the *Delete* button ( ) to remove the currently selected path from the paths list.
Link target to	Defines the target point of the camera. If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, the target can can be linked to either a point or a path. Select an option: Point: Links the target to a static point in the drawing. If *Link camera to* is set to *Point*, this option is not available for the target.  Do one of the following: • Click the list button, and choose a previously saved point. • Click the *Pick* ( ) button. You are prompted to specify a point in the drawing.

<table>
<tr>
<td></td>
<td>


Type a name for the point, then click the OK button.
The point is added to the points list.
• Click the Delete button (✕) to remove the currently selected point from the points list.
Path: Links a target to a linear entity in the drawing.


Do one of the following:
• Click the list button, and choose a previously saved path.
• Click the Pick ( ) button.
You are prompted to pick a linear entity in the drawing.
A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.


Type a name for the path, then click the OK button.
The path is added to the paths list.
• Click the Delete button (✕) to remove the currently selected path from the paths list.
</td>
</tr>
<tr>
<td>Animation Settings</td>
<td>
Allows to control the output of the movie file.


Frame rate (FPS): defines the number of frames per second. When you change the Frame rate, the Number of frames will be adjusted according to the Duration.
Number of frames: specifies the total number of frames in the movie. When you change the Number of frames, the Duration will be adjusted accordingly.
Duration (seconds): specifies the duration in seconds of the movie. When you change the Duration, the Number of Frames will be adjusted accordingly.
</td>
</tr>
</table>

	Reverse: Reverser the sequence of the frames. Corner deceleration: Slows down the camera at corners in the animation path. Visual style: Defines the visual style for the movie. Click the button, then select a visual style in the list. All visual styles in the current drawing are available. Resolution: Sets the width x height resolution of the movie. Click the button to select a resolution. Available resolutions are: 160x120, 320x240, 640x480, 800x600, 1024x768, 1280x1024, 1600x1024, 1920x1080, 1920x1200. Format: Specifies the file format of the movie. Available formats are: *avi*, *mpg* and *wmv*.
Preview	Displays the movement of the camera.
OK	Generates the movie. Displays a *File* dialog: • Select a folder. • Type a name for the movie. • (option) Select a file type, thus overriding the previously selected *Format* setting. • Click the *Save* button to create the movie file.

Related Commands

Camera - creates perspective views.
-Render - generates a photorealistic rendering using materials and lights through the command bar.
Render - generates a photorealistic rendering using materials and lights through a dialog box.
-VisualStyles - manages visual style definitions at the command line.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

AnnoReset

Resets all scale representations of the selected annotative entities to the location of their current scale representation.

Accessing the Command

command bar: *annoreset*

: ***annoreset***
Prompts you in the command bar:

Select annotative entities: **(Select entities)**
Entities in set: 1

Select annotative entities: **(Select more entities or right click)**
All scale representations of the selected annotative entities are reset to the location of the current scale representation.

Command Options

No options.

Related Commands

AnnoUpdate - updates the selected annotative entities to match their current style definition.

AnnoUpdate

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

Accessing the Command

command bar: *annoupdate*

: ***annoupdate***

Prompts you in the command bar:

Select annotative entities: **(Select entities)**
Entities in set: 1

Select annotative entities: **(Select more entities or right click)**
<x> found <y> were updated

Command Options

No Options

Related Commands

AnnoReset - resets all scale representations of the selected annotative entities to the location of their current scale representation.

Aperture

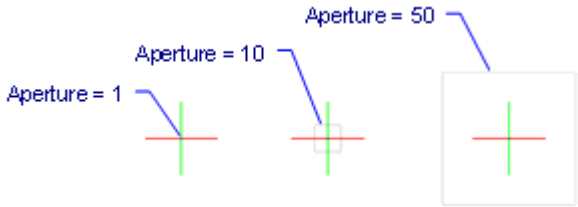
Sets selection area for snapping to entities.

Accessing the Command

command bar: *aperture*
transparent: *'aperture*
alias: *ap*
menu bar: *Settings | Entity Snap Precision*
toolbar: *Settings |*

: ***aperture***

Prompts you in the command bar:
Selection area for entity snap (1-50 pixels) <10>: (Enter a value between 1 and 50, or press Enter to keep the default value.)



The aperture cursor works in conjunction with entity snaps. BricsCAD searches the area inside the aperture cursor for entity features, such as endpoints and mid points.

A larger aperture cursor finds entity features more easily but can be too imprecise; a smaller one is more precise but can make it harder to find entity features.

By default, the aperture cursor is turned off. Use the ApBox system variable to turn it on.

Command Options

Option	Description
Selection area for entity snap	Changes the size of the aperture cursor; range is from 1 to 50 pixels wide.

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.

Apparent

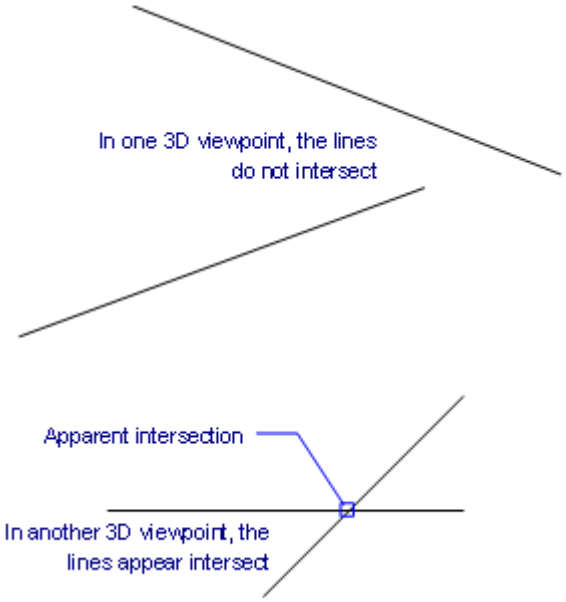
Toggles Apparent intersection entity snap; snaps to the intersections of entities, even when they only appear to intersect in 3D space.

Accessing the Command

command bar: *apparent*
transparent: *app*
alias: *2dintersection*
toolbar: *Entity Snaps* |

: ***apparent***

Displays no prompt in the command bar.



This entity snap snaps to intersecting entities, including those that appear to intersect in 3D space.

Command Option

Option	Description
Apparent	Entering this command toggles apparent intersection (2dInsersection) entity snap This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

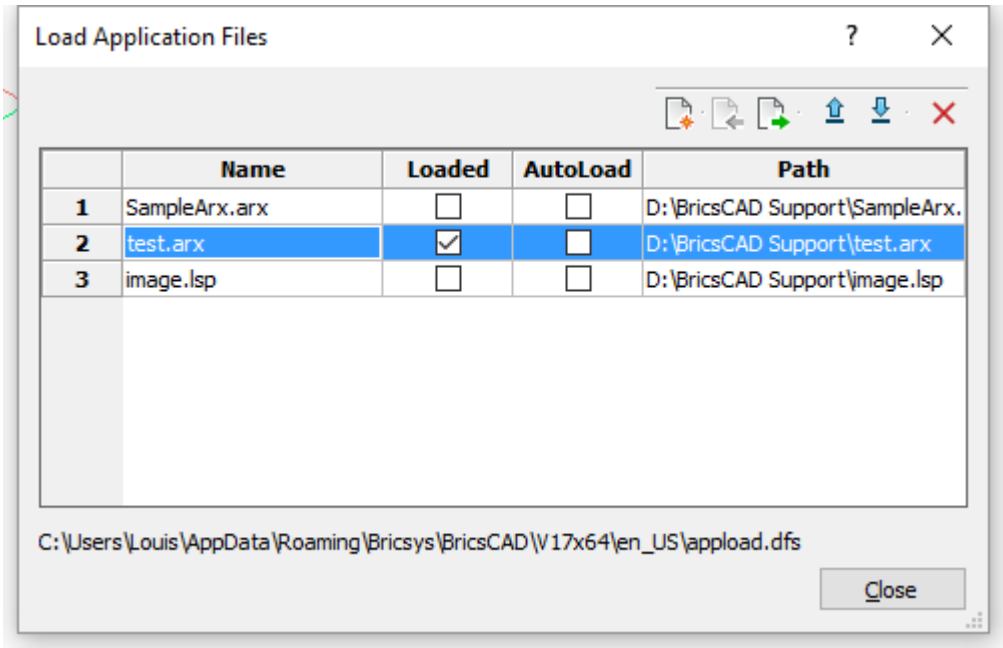
AppLoad

Loads DRX, LISP, and SDS applications to run inside BricsCAD.

Accessing the Command

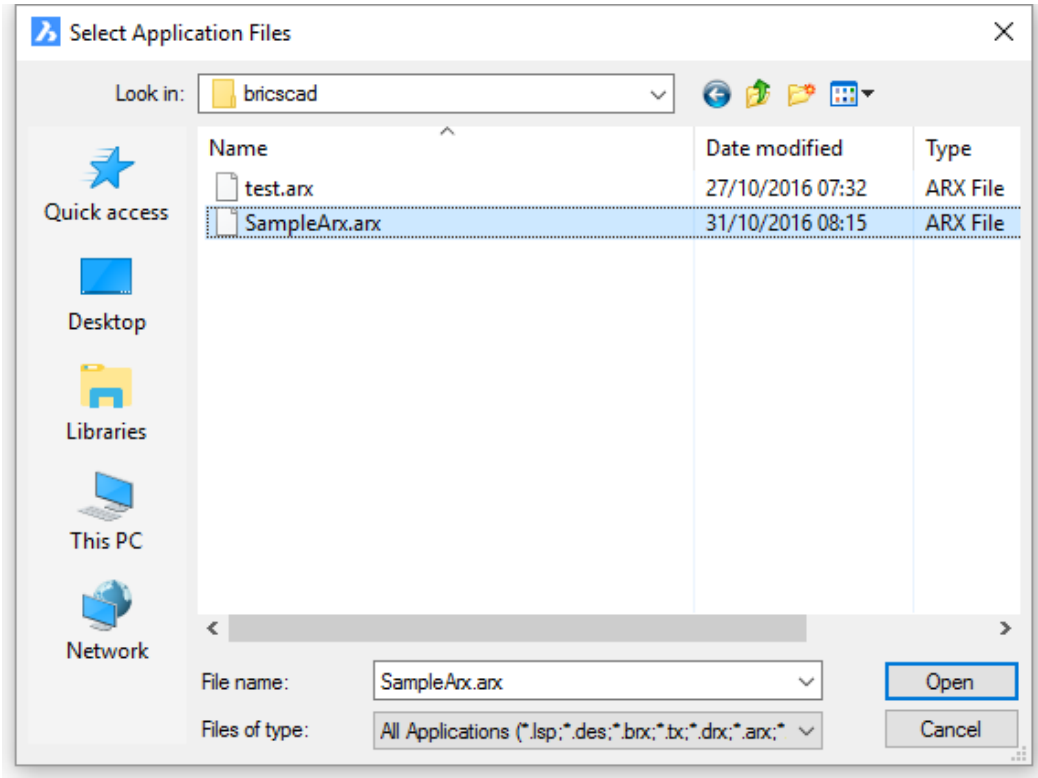
command bar: *appload*
menu bar: *Tools | Load Applications*

: ***appload***
Displays a dialog box:



Command Options

Option	Description
	Displays the *Select Application Files* dialog box. Optionally click the *Files of type* list button to select the file type of the applications to load. Select one or more files to load, then click the *Open* button.



	Loads the application into BricsCAD.
	Unloads the application from BricsCAD.
	Moves the selected application up in the list.
	Moves the selected application down in the list.
	Removes the selected application from the list.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
Commands - lists the names of internal and external commands.
CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

Arc

Draws arcs by a variety of methods.

Accessing the Command

command bar: *arc*
alias: *a*
menu bar: *Draw | Arc*
toolbar: *Draw |*

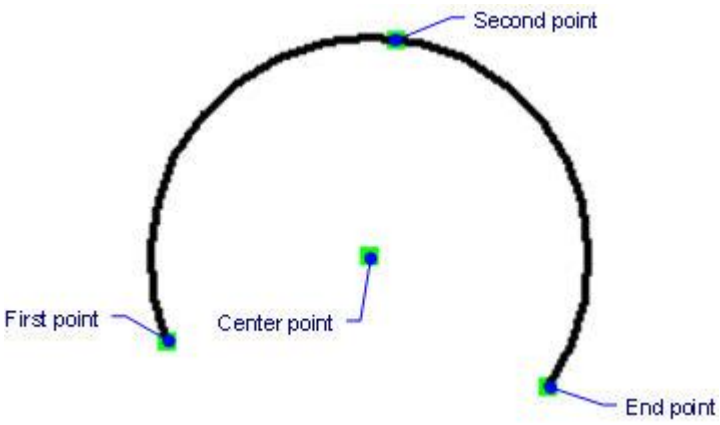
: ***arc***

Prompts you in the command bar:

Enter start of arc or [Center]: (Pick a point, enter coordinates, or enter an option.)

Enter second point or [Angle/Center/Direction/End/Radius]: (Pick a point, enter coordinates, or enter an option.)

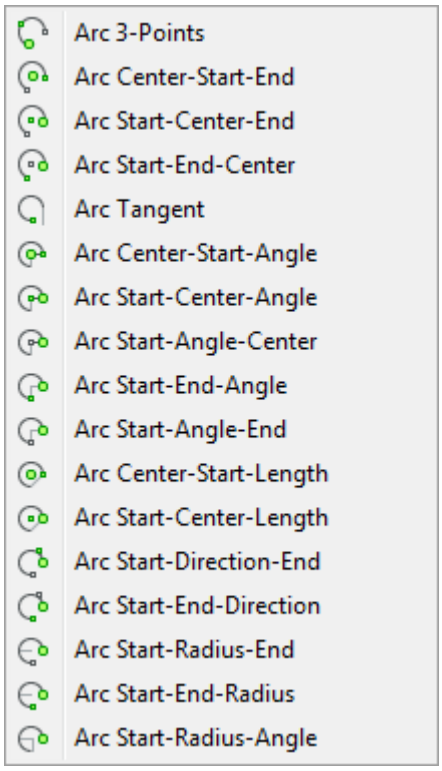
End point: (Pick a point or enter coordinates.)



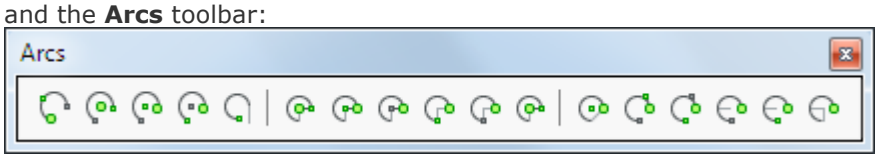
Subsequent uses of this command change the initial prompt to:

Enter start of arc or [Center/Follow] <Follow>: (Pick a point, enter coordinates, or enter an option.)

This command draws arcs by 17 methods. You can see the full list *in the* **Draw | Arc** *menu:*



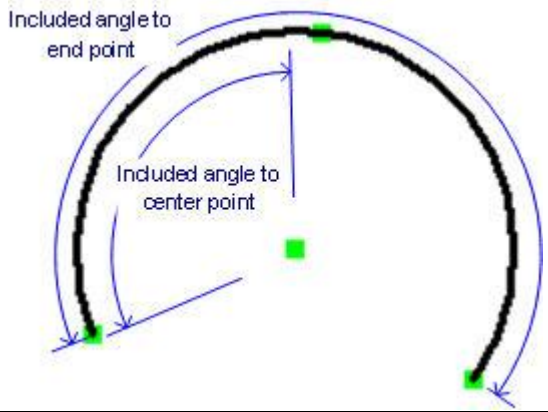
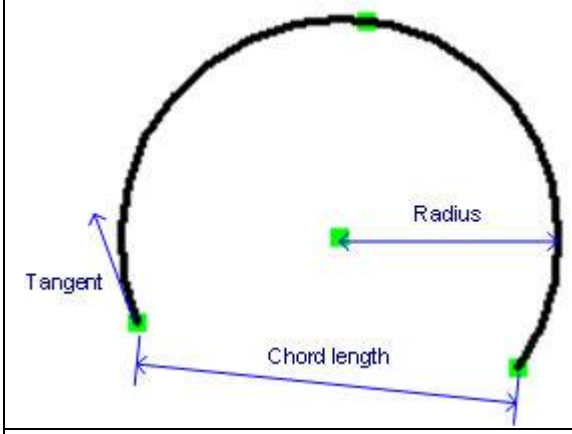
and the **Arcs** toolbar:



On each Arc icon a large green dot (●) indicates the first point, a small green dot (●) indicates the second point and a small white dot (○) indicates the last point.

Command Options

Option	Description
ENTER to use last point	Continues the arc tangent from the last point picked.
Center	Specifies the center point of the arc.
Follow	Continues the arc tangent from the last point picked (same as pressing Enter).
Start of arc	Specifies a different start point for the new arc.
Angle	Specifies the included angle, the angle between the arc's start point and: • Endpoint. • Center point.

	
Center 	Specifies the center point of the arc; after this, you can specify one of the following to complete the arc: • Endpoint. • Included angle. • Length of the chord.
Direction	Specifies the direction of the tangent to the arc's start point; following this, you specify the following: • End point.
End point	Specifies the arc's end point; following this, you can specify the following: • Center point. • Included angle. • Direction of tangency. • Radius.
Radius	Specifies the arc's radius; following this, you can specify the following: • Endpoint. • Included angle.
Second point	Specifies a point along the arc's circumference; following this, you specify the following: • Endpoint.

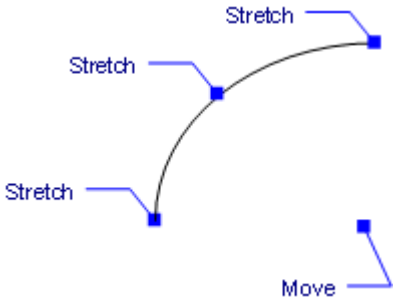
Procedures

General procedure to draw an arc
Editing an arc

Grips Editing

Arcs can be edited directly through grips:

1. Select the arc. Notice that it has four grips.
2. Drag the grips to perform the following actions:
    - Drag the endpoint and midpoint grips to stretch the arc.
    - Drag the center grip to move the arc.



Related Commands

Circle - draws arcs and circles.
Ellipse - draws elliptical arcs.
Join - joins arcs.
PEdit - converts arcs to polylines.
PLine - draws connected polyline arcs.
DimRadius and DimDiameter - dimensions arcs.
Change - changes the endpoints of arcs.

Center - snaps to the center of arcs.
Endpoint - snaps to the endpoints of arcs.
Quadrant - snaps to the 90-degree points on the circumference of arcs.
Tangent - snaps to the tangency of arcs.

Area

Finds the area and perimeter of 2D entities.

Accessing the Command

command bar: *area*
alias: *aa*
menu bar: *Tools | Inquiry | Area*
toolbar: *Inquiry |*

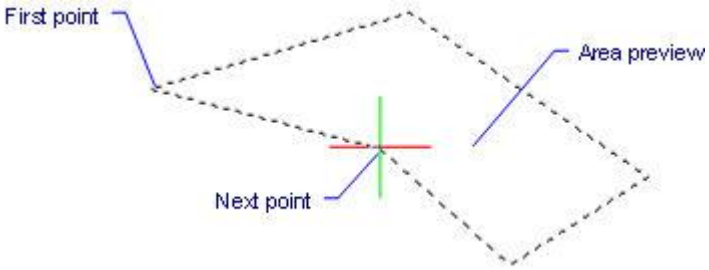
: ***area***

Prompts you in the command bar:
Select first point [**Entity**/**Add**/**Subtract**]: (Pick a point, or enter an option.)
<Next point>: *(Pick a point.)*

<Next point>: (Pick a point.)

<Next point>: (Pick a point or press Enter to stop.)

As you pick points, BricsCAD previews the area with a dashed polygon.



Use entity snap modes to ensure your picks are accurate.

When done picking points, press Enter; BricsCAD reports the area and perimeter like this:

Area = 6.1836 sq. units, Perimeter = 11.2968

This command does not work with solids (2D), traces, rays, xlines, or hatches. To find the area of hatches, use the Properties command.

Command Options

Option	Description
Entity	Provides the area and/or perimeter of the selected entity. The information reported by this command varies according to the entity selected: • Line and sketch report length. • Arc and elliptical arc report length. • Circle and ellipse report area and perimeter. • Closed polyline and closed spline report area and perimeter. • Open polyline and spline report area and perimeter as if they were closed. • Spline reports area and perimeter as if the polyline were closed. • 3D objects report the surface area.
Add	Allows you to work with two or more entities and/or areas. The prompt changes to: Adding: Entity/Subtract/ <First point> - pick a point, enter an option, or press Enter to exit Adding mode.

Subtract	Removes areas and/or entities from the total area and perimeter. The prompt changes to: Subtracting: Entity/Subtract/ <First point> - pick a point, enter an option, or press Enter to exit Subtracting mode.
First point	Specifies the first point of the area to be measured. The prompt changes to: <Next point> - pick another point, or press Enter to end the area specifying.

Related Commands

Dist - reports the 2D and 3D distances between two points.
Id - reports the x,y,z coordinates of picked points.
List - lists the area and other parameters of entities in the command bar.
MassProp - reports the properties of solid models and regions.
Properties - lists the area and other parameters of entities in a palette.

Array

Associatively distributes entity copies in an evenly spaced rectangular, polar or path array.

Accessing the Command

command bar: *array*
alias: *ar*
menu bar: *Modify | Array | 2D Array*
toolbar: *Modify |*

: ***array***

Prompts you in the command bar:

Select entities: (Select an entity)
Entities in set: 1

Select entities: (Select more entities or press Enter to stop selecting entities.)

Enter array type [Rectangular/PAth/POlar] <Rectangular>: ***(Choose an array type or press Enter to accept the default).***

The default array type is saved through the ARRAYTYPE system variable.

The ARRAYASSOCIATIVITY system variable controls whether static or associative arrays are created.

Command Options

Option	Description
Rectangular	Distributes entity copies into any number of rows, columns, and levels. See the ArrayRect command.
Path	Distributes entity copies evenly along a path into multiple rows, and levels. See the ArrayPath command.
Polar	Distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. See the ArrayPolar command.

Related Commands

3dArray - creates 3D arrays.
-Array - creates a polar or rectangular array of entities; displays prompts at the command bar.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
ArrayEdit - allows editing associative array entities and their source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.

-Array

Creates a static polar or rectangular array of entities; displays prompts at the command bar.

Accessing the Command

command bar: *-array*

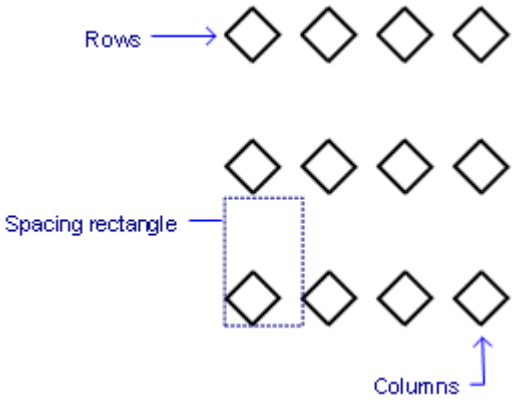
: ***-array***
Prompts you in the command bar:
Select entities to array: (Select one or more entities.)
Entities in set: 1
Select entities to array: (Press Enter to end entity selection.)
Type of array [Polar/Rectangular] <Rectangular>: ***(Enter an option.)***

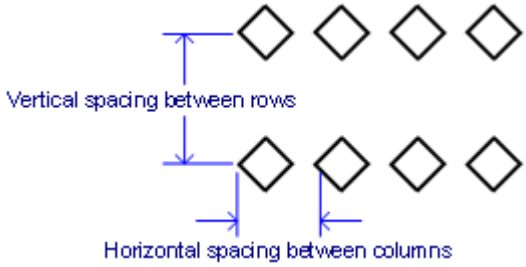
This command creates linear, rectangular, and polar arrays.

Command Options

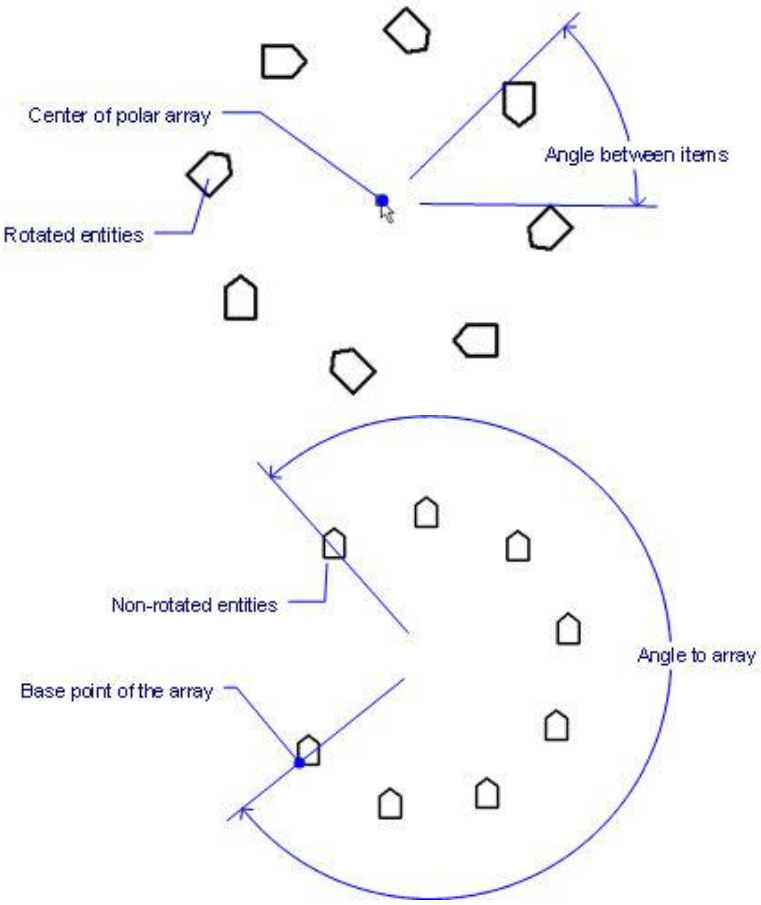
Option	Description
Select entities to array	Specifies the entities to be arrayed. Use any entity selection method; press Enter when you are done.
Polar	Creates polar (circular) arrays. Prompts you: **Center of polar array or [Base]: (Pick a point, or type B.)** **ENTER to specify angle between items/<Number of items to array>: (Enter a number, or press Enter.)** **Angle to fill (+ for ccw, - for cw)<360>: (Type a positive or negative value or press Enter to fill 360°.)** **Rotate entities around the array? [Yes-Rotate entities(Yes)/No-Do not rotate(No)]<Yes>: (Type N or Y.)**
Rectangular	Creates linear, rectangular, or square arrays. Prompts you: **Number of rows in the array <1>: (Enter the number of rows.)** **Number of columns <1>: (Enter the number of columns.)** **Vertical distance between rows, or spacing rectangle: (Specify the distance between rows, or pick two points.)** **Horizontal distance between columns: (Specify the distance between columns.)**

Rectangular arrays have rows and columns:



Rectangular Array Options	Description
Number of rows in the array	Specifies the number of rows; for a linear array, enter 1. To draw the array downwards, enter a negative number.
Number of columns	Specifies the number of columns. To draw the array to the left, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of entities.
Spacing rectangle	Specifies two points that indicate the spacing between rows and columns.
Horizontal distance between columns 	Specifies the distances between columns of entities.

Polar Array Options	Description
Base	Relocates the base point of the selected entities.
Center of polar array	Specifies the center point; enter coordinates or pick a point.
ENTER to specify angle between items	Specifies the angle between each copy.
Number of items to array	Specifies the number of items in the array; enter a number larger than 1

Angle to array	Determines the extent and direction of the array. Enter 360 for a full circle of copies; enter a smaller number to draw a partial polar array. Enter a positive angle to draw the array counter clockwise; enter a negative number to draw the array clockwise.
Rotate entities around the array	Determines whether copies are rotated when they are arrayed: • No does not rotate entities; all copies look the same as the original. • Yes rotates copies. 

Procedures

Creating a rectangular array
Creating a polar array

Related Commands

3dArray - creates 3D arrays.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
ArrayEdit - allows editing associative array entities and their source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.

ArrayClassic

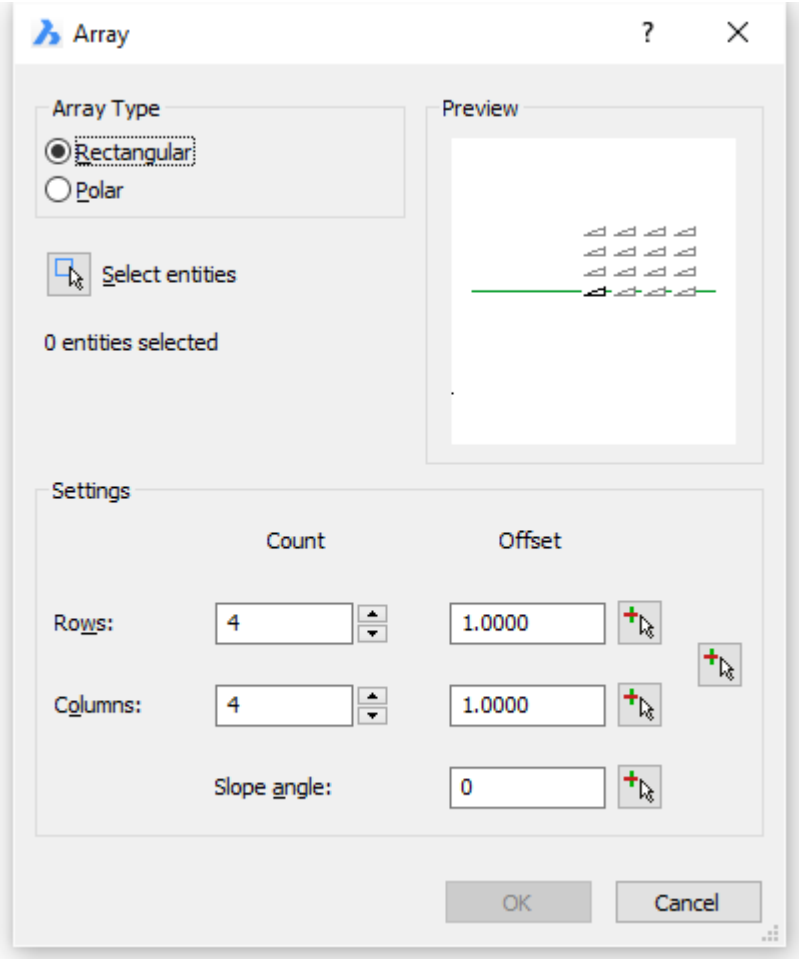
Creates a static rectangular or polar array of entities through a dialog box.

Accessing the Command

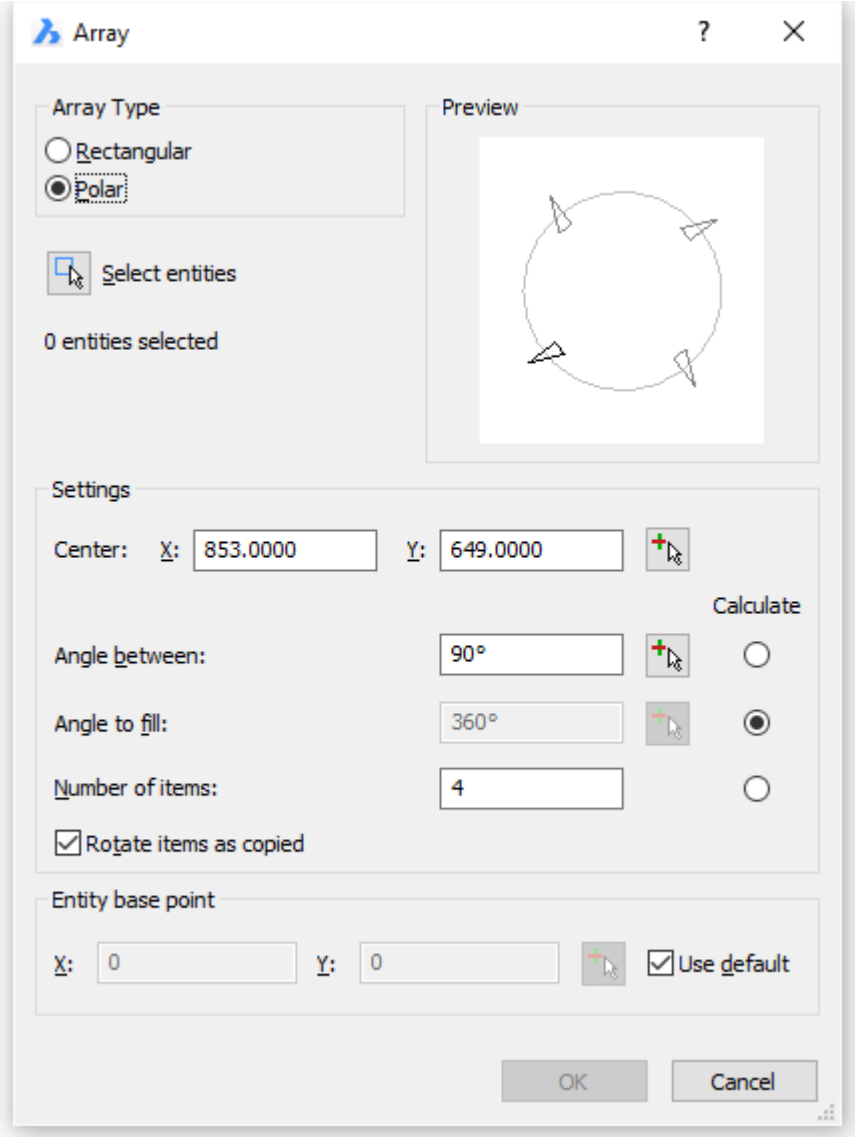
command bar: *arrayclassic*
menu: *Modify | Array | Array Dialog Box*

: ***arrayclassic***

Displays a dialog box: (Fill out the settings, then click the OK button to create the array.)



Rectangular array dialog box



Polar array dialog box

Command Options

Rectangular	**Rectangular Array command options**
Option	**Description**
Select entities	Click the *Select* button ( ) . The *Array* dialog box temporarily closes. You are prompted to select entities.
Rows Count	Specify the number of rows.
Rows Offset	Specify the offset between subsequent rows. A positive offset adds rows from left to right. Do one of the following: • Type a distance in the *Rows Offset* field.

	• Click the *Pick Points* button ( ). The *Array* dialog box temporarily closes. Prompts you: Vertical distance between rows: Specify two points in the drawing.
Columns Count	Specify the number of columns.
Columns Offset	Specify the offset between subsequent columns. A positive offset adds columns from bottom to top. Do one of the following: • Type a distance in the *Columns Offset* field. • Click the *Pick Points* button ( ). The *Array* dialog box temporarily closes. Prompts you: Horizontal distance between columns: Specify two points in the drawing.
Spacing Rectangle	Specify a rectangle. Rectangle length = Columns offset. Rectangle width = Rows offset. Click the *Pick Points* button ( ) The *Array* dialog box temporarily closes. Prompts you: Specify spacing rectangle: Specify two points in the drawing.
Slope Angle	Specify a rotation angle for the spacing rectangle. The angle is calculated with respect to the current coordinate system. Do one of the following: • Type an angle in the *Slope angle* field. • Click the *Pick Points* button ( ). The *Array* dialog box temporarily closes. Prompts you: Select angle of array: Specify two points in the drawing.
Polar	**Polar Array command options**
Option	**Description**
Select entities	Click the *Select* button ( ) . The *Array* dialog box temporarily closes. You are prompted to select entities.
Center	Specifies the center point of the array. Do one of the following: • Enter coordinates in the X and Y fields. • Click the *Pick Point* button ( ). The *Array* dialog box temporarily closes. Prompts you: Base/Center of polar array: Specify a point in the drawing.
Angle between	Specifies the angle between each copy. Do one of the following:

	• Click the *Calculate* radio button to calculate the angle from the *Angle to fill* and *Number of items*. • Type an angle in the *Angle* field. • Click the *Pick Points* button ( ). The *Array* dialog box temporarily closes. Prompts you: Specify angle between: A line displays dynamically from the current Center point. Specify a point in the drawing.
Angle to fill	Determines the extent and direction of the array. Enter 360 for a full circle of copies; enter a smaller number to draw a partial polar array. Enter a positive angle to draw the array counter clockwise; enter a negative number to draw the array clockwise. Do one of the following: • Click the *Calculate* radio button to calculate the angle from *Angle between* and *Number of items*. • Type an angle in the *Angle* field. • Click the *Pick Points* button ( ). The *Array* dialog box temporarily closes. Prompts you: Specify angle between: A line displays dynamically from the current Center point. Specify a point in the drawing.
Number of items	Specifies the number of items in the array Do one of the following: • Click the *Calculate* radio button to calculate the angle from the *Angle between* and *Angle to fill*. • Enter a number larger than 1.
Rotate items as copied	Determines whether copies are rotated when they are arrayed: • No does not rotate entities; all copies have the same orientation as the original. • Yes rotates copies.
Entity base point	Relocates the base point of the selected entities. Do one of the following: • Check the *Use Default* option (= lower left corner of the selection set bounding box). • Click the *Pick Point* button ( ). The *Array* dialog box temporarily closes. Prompts you: Base/Center of polar array: Specify a point in the drawing.

Procedures

Creating a static rectangular array
Creating a static polar array

Related Commands

3dArray - creates static rectangular and polar 3D arrays.
-Array - creates a static polar or rectangular array of entities; displays prompts at the command bar.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.

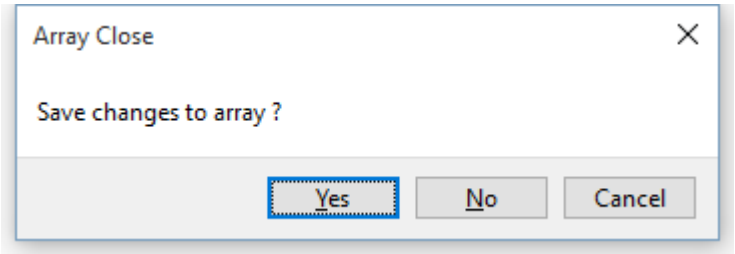
ArrayClose

Exits the array editing state and displays a dialog box to save or discard the changes made to the array source entities.
The command is only available when an associative array is checked out with ARRAYEDIT command.

Accessing the Command

command bar: *arrayclose*

: ***arrayclose***
Displays a dialog box:



Command Options

Option	Description
Yes	Saves the changes made to the source entities of the array and exits the array editing state.
No	Exits the array editing state and discards all changes, thus returning to the original state of the array.
Cancel	Closes the dialog box and stays in the array editing state.

Related Commands

ArrayEdit - allows editing associative array entities and their source entities.
-ArrayClose - Exits the array editing state and prompts you in the command bar to save or discard the changes made to the array source entities.

-ArrayClose

Exits the array editing state and prompts you in the command bar to save or discard the changes made to the array source entities.
The command is only available when an associative array is checked out with ARRAYEDIT command.

Accessing the Command

command bar: *-arrayclose*
Quad: *Modify|*

: ***-arrayclose***
Prompts you in the command bar:
Save changes to array? [Yes/No] <Yes>: *(Choose an option)*

Command Options

Option	Description
Yes	Saves the changes made to the source entities of the array and exits the array editing state.
No	Exits the array editing state and discards all changes, thus returning to the original state of the array.

Related Commands

ArrayEdit - allows editing associative array entities and their source entities.
ArrayClose - exits the array editing state and displays a dialog box to save or discard the changes made to the array source entities.

ArrayEdit

Allows editing associative array entities and their source entities.

Accessing the Command

command bar: *arrayedit*
menu bar: *Modify | Array | Edit Array*
toolbar: *Arrays |*

: ***arrayedit***
Prompts you in the command bar:

Select associative array: ***(Select an array entity.)***

Depending on the type of the selected array entity (rectangular, path or polar) the next prompt varies.

Rectangular array:

[**Source**/**REPlace**/**Base point**/**Rows**/**COLumns**/**Levels**/**RESet**/**eXit**]<eXit>: (Select an option.)

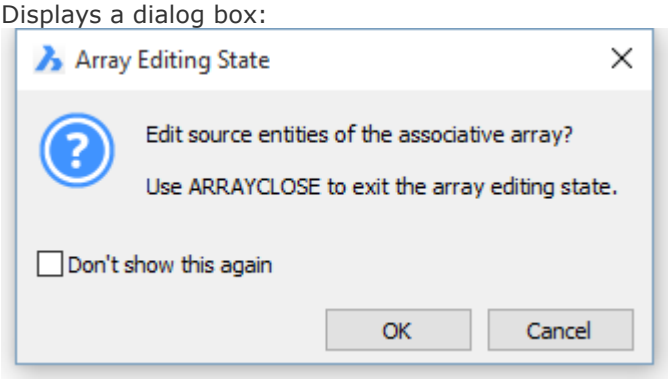
Path array:

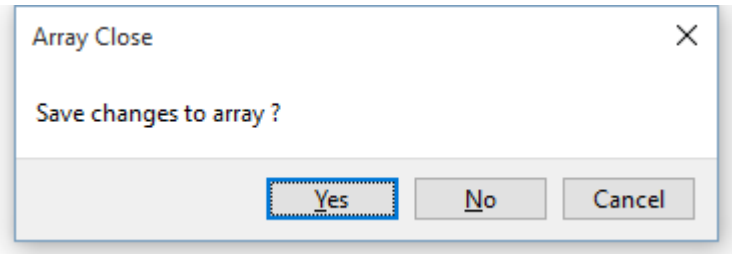
[**Source**/**REPlace**/**Method**/**Base point**/**Items**/**Rows**/**Levels**/**Align items**/**Z direction**/**RESet**/**eXit**]<eXit>: (Select an option.)

Polar array:

[**Source**/**REPlace**/**Base point**/**Items**/**Angle between**/**Fill angle**/**Rows**/**Levels**/**ROTate**/**RESet**/**eXit**]<eXit>: (Select an option.)

Command Options

Option	Description
Source	Allows to edit the source entities of the array. Prompts you: Select item in array: Select one of the items in the selected array entity. Displays a dialog box:  Click the *OK* button on the *Array Editing State* dialog box. Only the selected item in the array displays. Add, delete or modify entities. Type ARRAYCLOSE at the command prompt when done. A message box displays:

	 Click the *Yes* button. The other items of the array display. All changes are applied to all items in the array.
Replace	Allows to replace some or all items in the array. Prompts you: Select replacement objects: **Select one or more entities.** Entities in set: 1 Select replacement objects: Select more entities or right click. Enter base point of replacement objects or [Key point] <centroid>: Do one of the following: • Press Enter to use the centroid of the selection. • Specify a point. • Keypoint: prompts you: Select key point on replacement object as base point: When you move the cursor over the selected entities, all keypoints highlight one by one. Click to select a keypoint. Select an item in the array to replace or [Source objects]: Do one of the following: • Select one or more items in the array. • Source objects: replaces all items.
Reset	Restores erased entities and removes any item overrides.
Exit	Accept the changes and conclude the command.

Related Commands

3dArray - creates 3D arrays.
-Array - creates a polar or rectangular array of entities; displays prompts at the command bar.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
-ArrayClose - exits the array editing state and prompts you in the command bar to save or discard the changes made to the array source entities.
ArrayClose - exits the array editing state and displays a dialog box to save or discard the changes made to the array source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.

ArrayPath

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

Accessing the Command

command bar: *arraypath*
menu bar: *Modify | Array | Path Array*
toolbar: *Modify | Arrays |*
ribbon:
Sketch | Arrange | Path Array (Mechanical and SheetMetal Workspace)
Solid | Edit | Path Array (Mechanical and SheetMetal Workspace)
Model | Modify 3D | Path Array (BIM Workspace)
Change | Arrange | Path Array (BIM and 3D Modeling Workspace)

: ***arraypath***
Prompts you in the command bar:

Select entities to array: **(Select an entity.)**

Entities in set: 1

Select entities to array: (Select more entities or press Enter.)

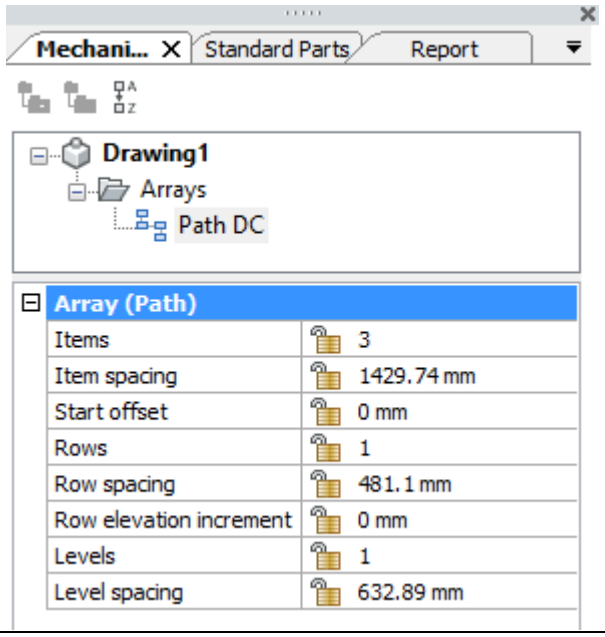
Type = Path, Associative = Yes

Select path curve: **(Select a linear entity: line, polyline, arc, circle, ellipse, elliptical arc, spline, helix or 3D polyline.)**

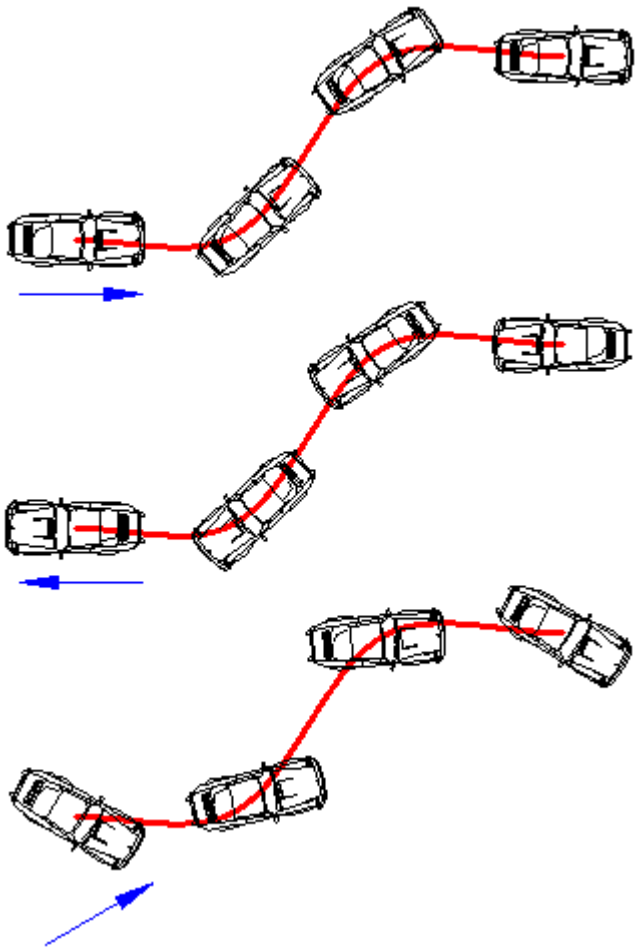
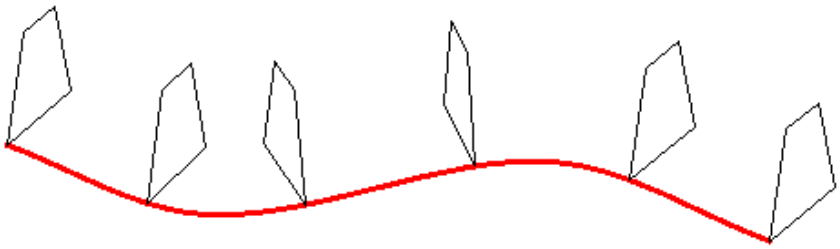
[ASsociative/Method/Base point/Tangent direction/Items/Rows/Levels/Align items/Z direction/eXit]<eXit>: **(Press Enter to accept the current settings or select an option.)**

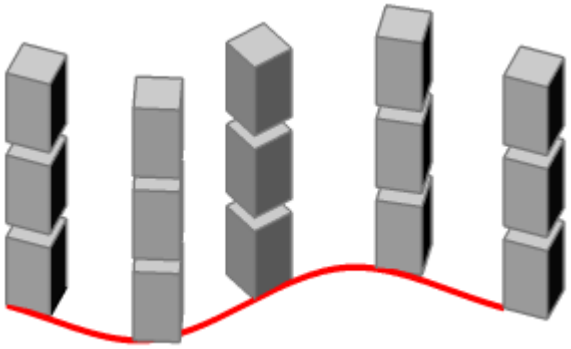
Command Options

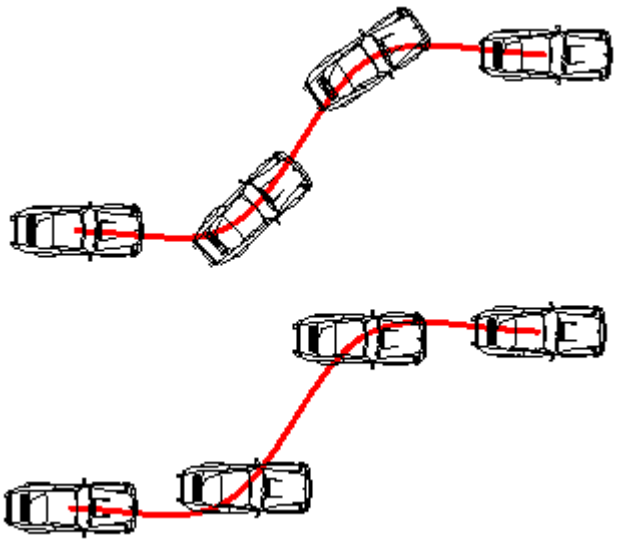
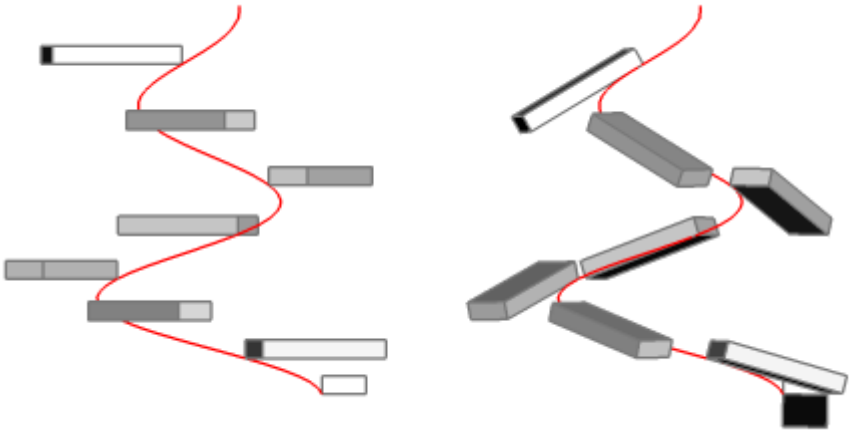
Option	Description
Associative	Determines whether an array entity is created or a series of copies of the selected entities. The properties of array entities can be modified using the ARRAYEDIT command. The default setting is saved by the ARRAYASSOCIATIVITY system variable. Prompts you: Create associative array? [Yes/No]<Yes>: • **Yes**: An Array entity is created. • **No**: A series of copies of the selected entities is created. The associative arrays parameters can be specified using a parametric expressions. Expressions can be set in the array properties in bottom part of the Mechanical Browser, Properties panel or via the ARRAYEDIT command.



Method	Specifies how the selected entities are distributed along the path. Prompts you: Enter path method or [Divide/Measure]<Measure>: • **Divide**: Distributes a specified number of items evenly along the path. • **Measure**: Places the items at a specified interval.
Base point	Sets the base point of the array. Items are placed relative to the base point. Prompts you: Enter base point: Specify a point in the drawing.
Tangent direction	Determines how the items are aligned relative to the starting direction of the path. Prompts you: Select first point of tangent direction vector or [Normal]: • **2 points**: Specify two points to 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.

	 • **Normal**: Orients the Z direction of the first item with the starting direction of the path 
Items	Specifies the number of items or the distance between items, depending on the Method setting. Method = Divide: Prompts you: **Enter number of items along path <xxx>:** Specify the number of items in the array. Method = Measure: Prompts you: Enter the distance between items along path <xxx: Specify a distance*. Maximum items = xxx

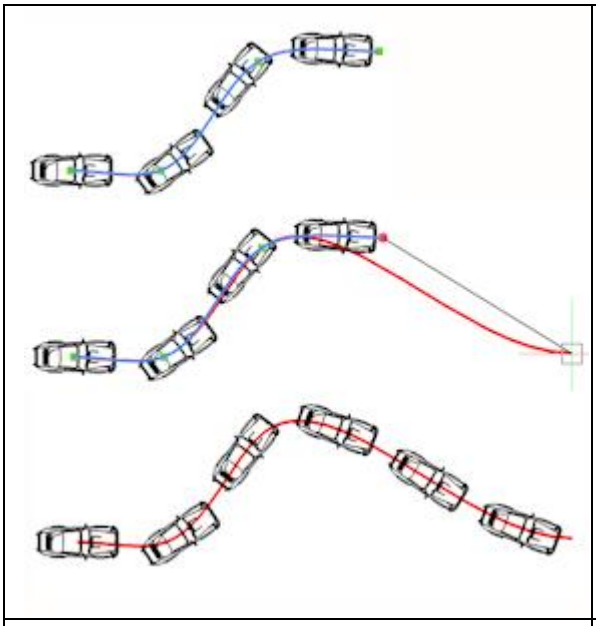
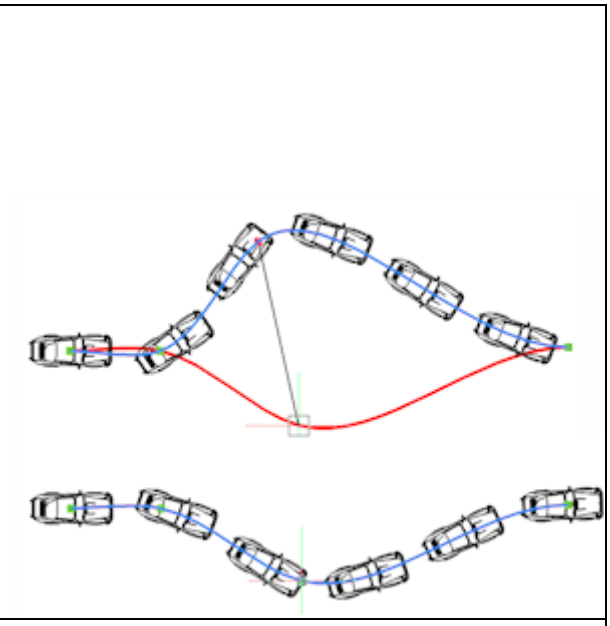
	Enter number of items or [Fill entire path] <xxx>: **Press Enter or choose an option:** • Enter a number of Items: **By default the items are placed along the entire path at the specified distance. This option allows to specify a smaller number if you only want to place items along a portion of the path.** • Fill entire path: **The number of items is adjusted if the length of the path is modified (see: Grips Editing: Editing the path length).**
Rows	Specifies the number of rows in the array, the distance between them and the incremental elevation. Prompts you: Enter the number of rows <1>: Specify the number of rows. Enter the distance between rows or [Total] <1.0>: • **Distance**: specify the distance* between subsequent rows. • **Total**: specify the distance* between the first and the last row. Enter the incrementing elevation between rows <0.0>: Defines the increasing or decreasing elevation for each subsequent row. Specify a value.
Levels	Allows to create 3D arrays. Specifies the number and spacing of levels.  Prompts you: Enter the number of levels <1>: Specify the number of levels. Enter the distance between levels or [Total] <xxx>: • **Distance:** Specify the distance* between subsequent levels. • **Total:** Specify the distance* between the bottom and the top level.
Align items	Specifies whether to align each item to be tangent to the path direction. Alignment is relative to the first item's orientation. Prompts you: Align arrayed items to path? <Yes>/No • **Yes**: Each item is placed tangent to the path direction. (top)

	• **No**: Each item keeps the orientation of the first item. (bottom) 
Z direction	Controls whether to maintain the items' original Z direction or to naturally bank the items along a 3D path. Prompts you: Maintain Z direction for all items in the array? [Yes/No]<Yes>: • **Yes**: Z-direction (vertical) is maintained (left). • **No**: Items banked along the helix (right). 
Exit	Accept the current settings and conclude the command.

* To specify a distance: type a value or specify two points.

Grips Editing

When grips editing the path, the array entity is recreated along the modified path.

	
Editing the path length.	*Editing the path shape.*

Related Commands

3dArray - creates static rectangular and polar 3D arrays.
-Array - creates a static polar or rectangular array of entities; displays prompts at the command bar.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
ArrayEdit - allows editing associative array entities and their source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.
-BmParameters - allows to list and edit the parameters of inserted components and associative array.

ArrayPolar

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

Accessing the Command

command bar: *arraypolar*
menu bar: *Modify | Array | Polar Array*
toolbar: *Modify | Arrays |*
ribbon:
Sketch | Arrange | Path Array (Mechanical and SheetMetal Workspace)
Solid | Edit | Path Array (Mechanical and SheetMetal Workspace)
Model | Modify 3D | Path Array (BIM Workspace)
Change | Arrange | Path Array (BIM and 3D Modeling Workspace)

: ***arraypolar***
Prompts you in the command bar:

Select entities to array: (Select an entity.)

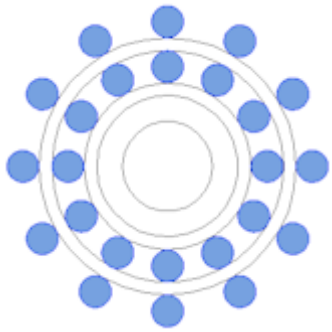
Entities in set: 1

Select entities to array: (Select more entities or press Enter.)

Type = Polar, Associative = Yes

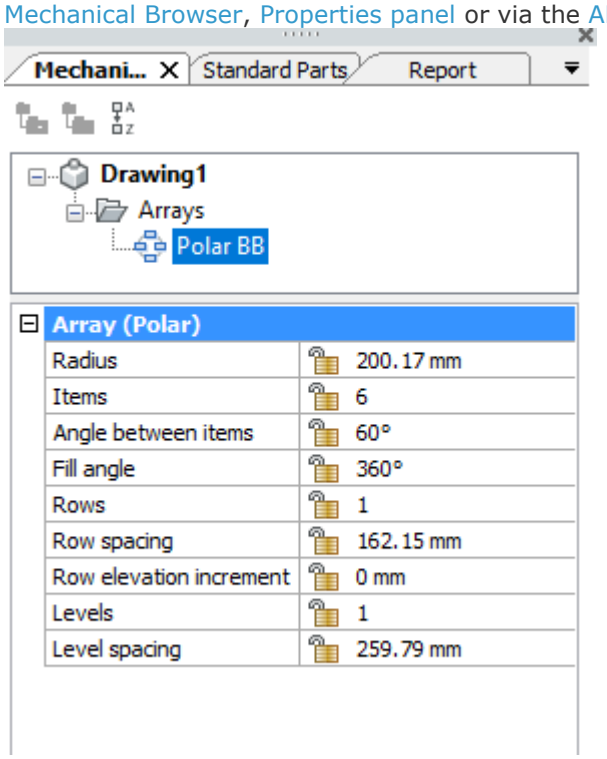
Enter center point of array or [Base point/Axis of rotation]: **(Specify a point or choose an option.)**

[ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate/eXit]:<eXit>: **(Press Enter to accept the current settings or select an option.)**

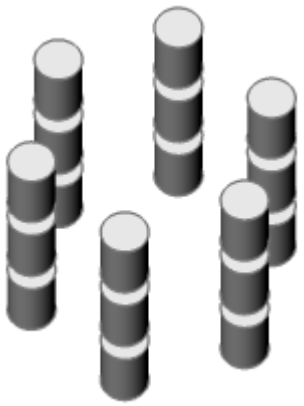


Command Options

Option	Description
Center point	Specifies the center point of the polar array. The axis of rotation is parallel to the Z-axis of the current coordinate system.
Axis of rotation	Allows to specify the axis of rotation. Prompts you to specify two points.
Associative	Determines whether an array entity is created or a series of copies of the selected entities. The properties of array entities can be modified using the ARRAYEDIT command. The default setting is saved by the ARRAYASSOCIATIVITY system variable. Prompts you: Create associative array? [Yes/No]<Yes>:

	• Yes: An Array entity is created. • No: A series of copies of the selected entities is created. The associative arrays parameters can be specified using a parametric expressions. Expressions can be set in the array properties in bottom part of the Mechanical Browser, Properties panel or via the ARRAYEDIT command. 
Base point	Sets the base point of the array. Items are placed relative to the base point. Prompts you: Enter base point: Specify a point in the drawing.
Items	Specifies the number of items in the array. Prompts you: Enter number of items in array <xx>: Type a value or press Enter to accept the current number.
Angle between	Specifies the angle between two subsequent items. Prompts you: Enter angle between items <xx.xx>: Type a value or press Enter to accept the current angle.
Fill Angle	Specifies the angle between the first and the last item. Prompts you: Enter the angle to fill (+=ccw, -=cw) <xx.xx>: Type a value or press Enter to accept the current angle.
Rows	Specifies the number of rows and the spacing between subsequent rows. Prompts you: **Enter the number of rows <xx>:** Type a value or press Enter to accept the current number. Enter the distance between rows or [Total] <xx.x>: • Distance: specify the distance* between subsequent rows. • Total: specify the distance* between the first and the last row. Enter the incrementing elevation between rows <0.0>:

	Defines the increasing or decreasing elevation for each subsequent row. Specify a value.

Levels	Allows to create 3D arrays. Specifies the number and spacing of levels.  Prompts you: **Enter the number of levels <1>:** Specify the number of levels. Enter the distance between levels or [Total] <xxx>: • Distance: Specify the distance* between subsequent levels. • Total: Specify the distance* between the bottom and the top level. * To specify a distance: type a value or specify two points.
Rotate	Controls whether items in the array are rotated. Prompts you: Rotate arrayed items? <Yes>/No: • Yes: Items are rotated. • No: The orientation of the source entities is preserved.
Exit	Accept the current settings and conclude the command.

Related Commands

3dArray - creates static rectangular and polar 3D arrays.
-Array - creates a static polar or rectangular array of entities; displays prompts at the command bar.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
ArrayEdit - allows editing associative array entities and their source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
MInsert - inserts arrays of blocks.
-BmParameters - allows to list and edit the parameters of inserted components and associative array.

ArrayRect

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

Accessing the Command

command bar: *arrayrect*
menu bar: *Modify | Array | Rectangular Array*
toolbar: *Modify | Arrays |*
ribbon:
Sketch | Arrange | Path Array (Mechanical and SheetMetal Workspace)
Solid | Edit | Path Array (Mechanical and SheetMetal Workspace)
Model | Modify 3D | Path Array (BIM Workspace)
Change | Arrange | Path Array (BIM and 3D Modeling Workspace)

: ***arrayrect***
Prompts you in the command bar:

Select entities to array: ***(Select an entity.)***

Entities in set: 1

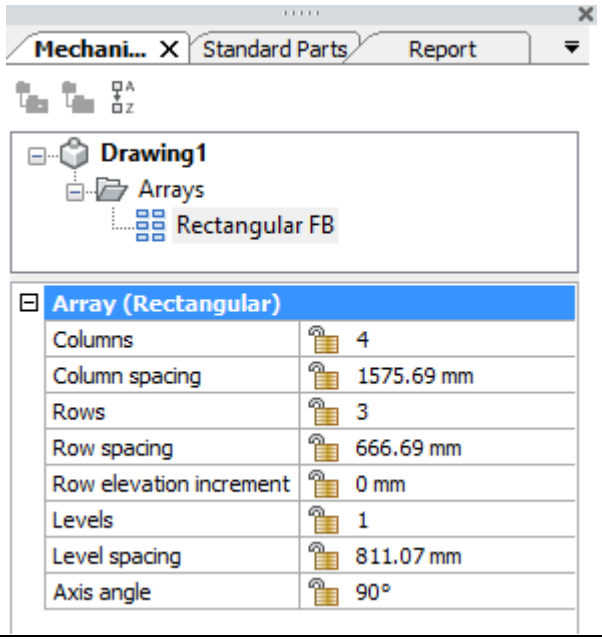
Select entities to array: (Select more entities or press Enter.)

Type = Rectangular, Associative = Yes

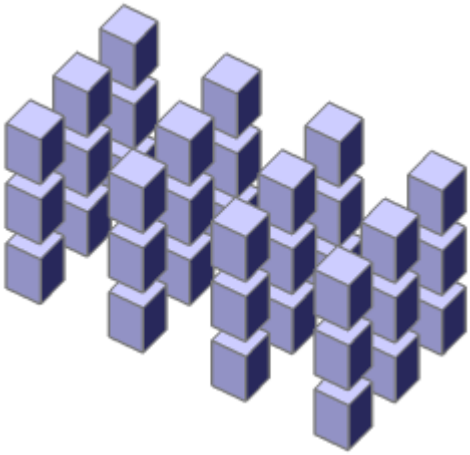
[**ASsociative**/**Base point**/**COUnt**/**Spacing**/**COLumns**/**Rows**/**Levels**/**eXit**]<eXit>: *(Press Enter to accept the current settings or select an option.)*

Command Options

Option	Description
Associative	Determines whether an array entity is created or a series of copies of the selected entities. The properties of array entities can be modified using the ARRAYEDIT command. The default setting is saved by the ARRAYASSOCIATIVITY system variable. Prompts you: Create associative array? [Yes/No]<Yes>: • Yes: An Array entity is created. • No: A series of copies of the selected entities is created. The associative arrays parameters can be specified using a parametric expressions. Expressions can be set in the array properties in bottom part of the Mechanical Browser, Properties panel or via the ARRAYEDIT command.

	
Base point	Sets the base point of the array. Items are placed relative to the base point. Prompts you: Enter base point: Specify a point in the drawing.
Count	Specifies the number of rows and columns. Prompts you: Enter the number of columns <xx>: Type a value or press Enter to accept the current number. Enter the number of rows <xx>: Type a value or press Enter to accept the current number.
Spacing	Allows to specify the spacing between columns and/or rows. Prompts you: Enter the distance between columns or [Unit cell] <xx.x>: Type a value or press Enter to accept the current spacing. Option: Type U + Enter or choose *Unit Cell* in the prompt menu. Prompts you: Select first point of unit cell: Specify a point. A rectangle displays dynamically. If Dynamic Input = ON, the width and height of the spacing rectangle display. Opposite corner: Do one of the following: • Specify a point. • Type the desired column and row spacing in the dynamic entry fields. Press the TAB key to go back and forth between the entry fields. Press Enter to confirm the current values.
Columns	Specifies the number of columns and the spacing between subsequent columns. Prompts you: **Enter the number of columns <xx>:** Type a value or press Enter to accept the current number. Enter the distance between columns or [Total] <xx.x>:

	• Distance: specify the distance* between subsequent columns. • Total: specify the distance* between the first and the last column.

Rows	Specifies the number of rows and the spacing between subsequent rows. Prompts you: **Enter the number of rows <xx>:** Type a value or press Enter to accept the current number. Enter the distance between rows or [Total] <xx.x>: • Distance: specify the distance* between subsequent rows. • Total: specify the distance* between the first and the last row. Enter the incrementing elevation between rows <0.0>: Defines the increasing or decreasing elevation for each subsequent row. Specify a value.
Levels	Allows to create 3D arrays. Specifies the number and spacing of levels.  Prompts you: Enter the number of levels <1>: Specify the number of levels. Enter the distance between levels or [Total] <xxx>: • Distance: Specify the distance* between subsequent levels. • Total: Specify the distance* between the bottom and the top level.
Exit	Accept the current settings and conclude the command.

* To specify a distance: type a value or specify two points.

Related Commands

3dArray - creates static rectangular and polar 3D arrays.
-Array - creates a static polar or rectangular array of entities; displays prompts at the command bar.
Array - associatively distributes entity copies in an evenly spaced rectangular, polar or path array.
ArrayClassic - creates a static polar or rectangular array of entities through a dialog box.
ArrayEdit - allows editing associative array entities and their source entities.
ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.

MInsert - inserts arrays of blocks.
-BmParameters - allows to list and edit the parameters of inserted components and associative array.

AttDef

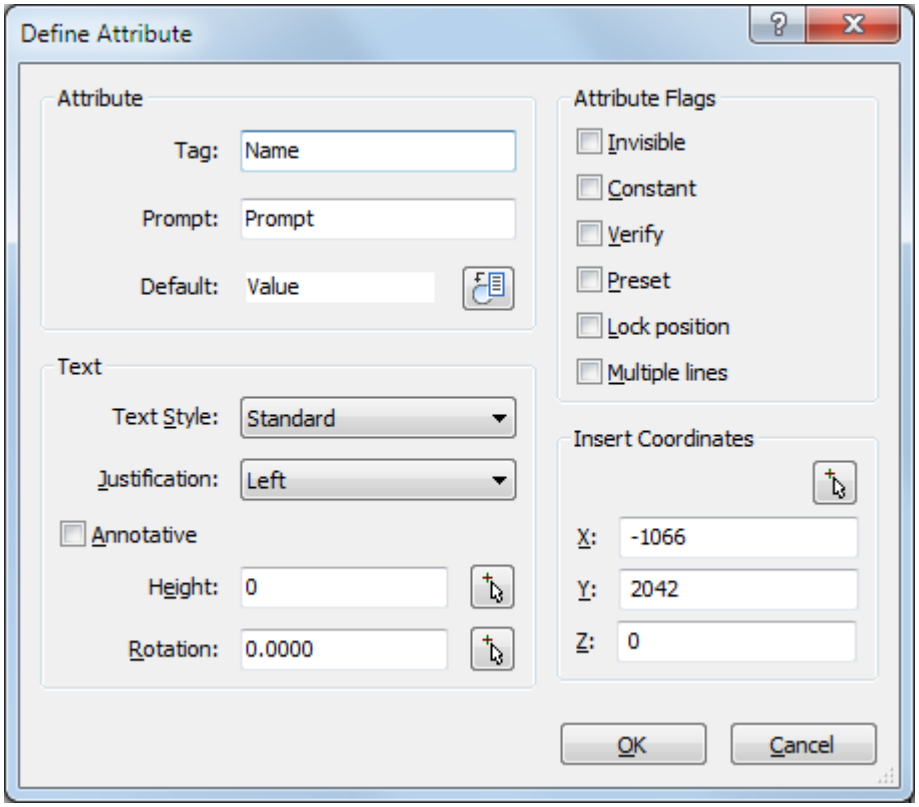
Defines how a block's attributes will be created through a dialog box.

Accessing the Command

command bar: *attdef*
aliases: *at, ddattdef*
menu bar: *Tools | Attributes | Define Attributes*
toolbar: *Attributes |*

: ***attdef***

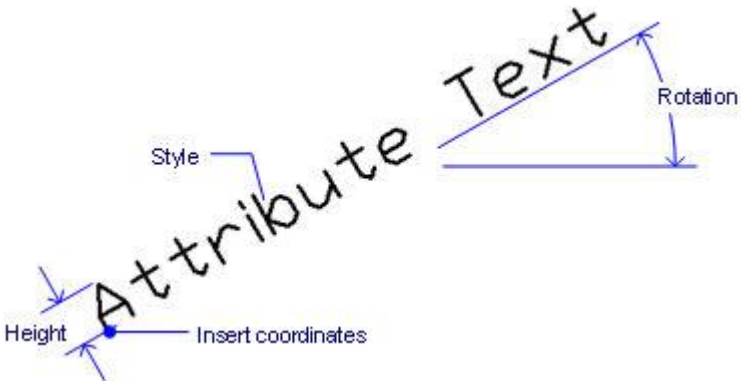
Displays a dialog box:



*Enter options, and then click **OK**.*

To create and use attributes, follow these steps:

1. Use this command to define attributes.
2. Use the **Block** command to combine attributes and symbols as blocks.
3. Use the **Insert** command to insert blocks with attributes.
4. Use the **AttEdit** command to edit the values of attributes, if necessary; use the **AttDisp** command to toggle the display of attributes in drawings.
5. Use the **AttExt** command to extract attribute values to external data files.



Command Options

Option	Description
Tag	Specifies the name of the attribute; this is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.
Prompt	Specifies the user prompt; this is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank; BricsCAD uses the tag as the prompt during attribute insertion.
Default	Specifies the default value; this is displayed in angle brackets, like <360>. As with other defaults, the user need only press Enter to accept this value. Click the *Insert Field* button ( ) to assign a field value to the attribute (see the FIELD command).
Text	
Text Style	Specifies the text style for the attribute text; choose from one of the styles defined in the drawing. (You can create additional text styles with the Style command.)
Justification	Specifies the justification of the attribute text, such as Center or Fit.
Height	Specifies the height of the attribute text, unless overridden by the text style. Enter a value, or click the button.
Annotative	Sets the *Annotative* property of the attribute text.
Height	Dismisses the dialog box temporarily, and then prompts you at the command prompt: **Select text height** - enter a value, or pick two points in the drawing.
Rotation	specifies the rotation angle of the rotation text. Enter a value, or click the button.
Rotation	Dismisses the dialog box temporarily, and then prompts you at the command prompt: **Select text angle** - enter an angle, or pick two points in the drawing.
Attribute Flags	
Invisible	Hides the attributes; they are not displayed and not printed. (Hidden attributes can be displayed with the AttDisp command.)
Constant	Uses the default value; the user cannot change the value.
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.
Lock Position	Locks the position of the attribute within the block reference. Unlocked attributes can be moved relative to the rest of the block using grip editing and multiline attributes can be resized.
Multiple lines	Uses multiline text instead of single line text. If the *Multiple lines* flag is checked, a button (...) 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. Prompts you: **Specify location of multiline attribute:** specify a point. The MTEXT command is launched. Type the default text for the attribute, then click the *OK* button on the *Text Formatting* toolbar.
Insert Coordinates	
Insert Coordinates	Dismisses the dialog box temporarily, and then prompts you at the command prompt: **Select insertion point** -enter coordinates, or pick a point in the drawing.
X	Specifies the x coordinate of the attribute's insertion point.
Y	Specifies the y coordinate.
Z	Specifies the z coordinate.

Procedures

Defining attributes

Related Commands

-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
DdAtte - edits the values of attributes through a dialog box.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Field - inserts a field in a multiline text, table cell or attribute.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.

RefEdit - edits blocks and attributes in-place.

-AttDef

Defines how a block's attributes will be created at the command prompt.

Accessing the Command

command bar: *-attdef*

: ***-attdef***

Prompts you in the command bar:

Toggle attribute modes: Constant=Off Invisible=Off Preset=Off Verify=Off Lock position=Off Multiple lines=Off <done>: Enter an option, or press Enter.)

Name of attribute: (Enter up to 255 characters.)

Prompt: (Enter up to 255 characters.)

Default text: (Enter up to 255 characters.)

Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point, or enter an option.)

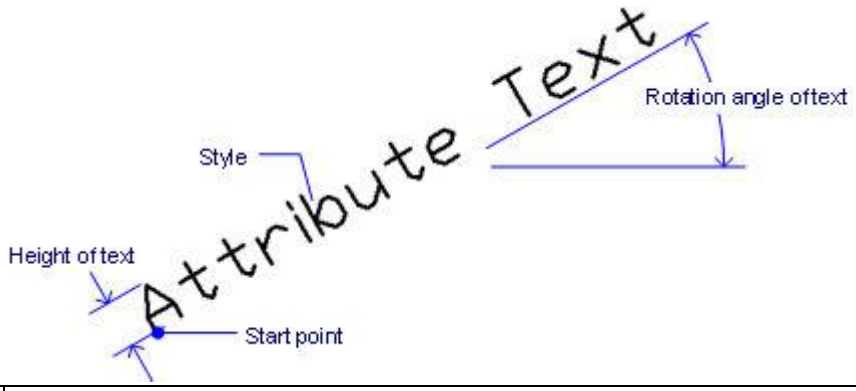
Height of text <0.2000>: (Enter a value, or press Enter to accept the default.)

Rotation angle of text <0>: (Enter a value, or press Enter to accept the default.)

This command is meant for use by macros, scripts, and LISP routines.

Command Options

Option	Description
Toggle attribute modes	Sets modes for the attributes:
Constant=Off	If On, uses the default value; the user cannot change the value. Type *C*, then press Enter to toggle the attribute mode.
Invisible=Off	If On, hides the attribute; hidden attributes are not displayed nor printed. (Hidden attributes can be displayed with the AttDisp command.) Type *I*, then press Enter to toggle the attribute mode.
Preset=Off	If On, inserts attributes without prompting the user; the attributes can be changed later with the AttEdit command. Type *P*, then press Enter to toggle the attribute mode.
Verify=Off	If On, forces the user to enter the value a second time; this helps to ensure the value is entered correctly. Type *V*, then press Enter to toggle the attribute mode.
Lock position=Off	If On, locks the position of the attribute within the block reference. Unlocked attributes can be moved relative to the rest of the block using grip editing and multiline attributes can be resized. Type *L*, then press Enter to toggle the attribute mode.
Multiple lines=Off	If on, uses multiline text instead of single line text. Type *M*, then press Enter to toggle the attribute mode.
<done>	Press Enter to continue with the remainder of the command.

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; this is displayed in angle brackets, like <360>. As with other defaults, the user need only press Enter to accept this value. If the *Multiple lines* mode is ON, you are prompted: Next line or <done>: Type the next line of the Mtext or press Enter to stop.
Text	Specifies the style and justification of the text: • Style specifies the text style; enter the name of a style present in the drawing. • Align aligns the text between two points; the text height is adjusted to maintain its aspect ratio. • Fit aligns the text between two points; the text height is maintained. • Center centers the text's baseline on the pick point. • Middle centers the text on the pick point. • Right right justifies the text. • Justify accesses additional justification options. • Start point left justifies the text.
	
Height of text	Specifies the height of the attribute text, unless overridden by the text style or justification. Enter a value, or pick two points to indicate the height.
Rotation angle of text	Specifies the rotation angle of the rotation text. Enter a value, or pick two points to indicate the angle relative to the positive x axis.

Related Commands

AttDef - defines attributes through a dialog box.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

AttDisp

Shows all attributes, none, or those normally visible.

Accessing the Command

command bar: *attdisp*
transparent: *'attdisp*
alias: *ad*
menu bar: *Tools | Attributes | Attribute Display*
toolbar: *Attributes |*

: ***attdisp***
Prompts you in the command bar:
Attribute display mode [ON/OFf/Normal] <Normal>: *(Enter an option.)*
When attributes are off, they do not display in the drawing and are not plotted.

Command Options

Option	Description
Attribute display mode	Changes the display of attribute values: • ON displays all attributes, including ones set to invisible mode. • OFF hides all attributes. • Normal displays attributes, except those set to invisible mode.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

AttEdit

Edits attributes' values and properties.

Accessing the Command

command bar: *attedit*
alias: *-ate*
menu bar: *Tools | Attributes | Edit Attributes*
toolbar: *Attributes |*

: ***attedit***

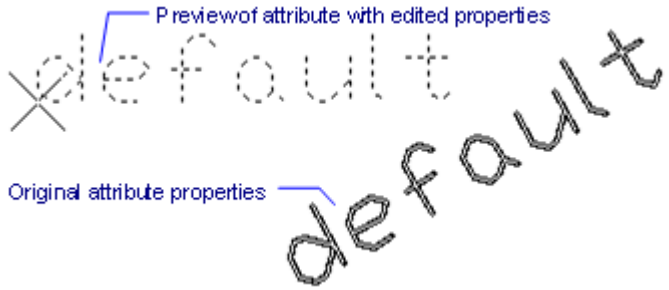
Prompts you in the command bar:
Edit attributes one at a time? [Yes/No] <Yes>: (Enter Y or N.)

- Y (yes) edits attributes one at a time; allows changes to values and properties.
- N (no) edits attributes globally; allows changes to values only.

This command is meant for editing attributes globally, and for use with macros, scripts, and LISP routines.

For editing individual attributes, it is easier to use the dialog box of the EAttEdit command.

As you change the attribute text, BricsCAD previews them:



To edit the values and all properties of attribute text one attribute at a time:

Edit attributes one at a time? <Y> (Press Enter for yes.)

Edit attributes of which blocks? <*>: (Press Enter for all, or specify the block names.)

Edit attributes with these names <*>: (Press Enter for all, or specify the attribute tags.)

Edit attributes with this text <*>: (Press Enter for all, or specify the attribute values.)

Select attributes: (Select one or more attributes.)

Entities in set: 1

Select attributes: (Press Enter to end entity selection.)

1 attributes were selected.

[Position/Angle/Text/Style/Color/Height/Layer/Next/pRev/Quit]<Next>: ***(Enter an option, or press Enter to exit the command.)***

To edit the values only of all attributes:

Edit attributes one at a time? <Y>: (Type N.)

Global edit of attribute values.

Edit only attributes visible on screen? <Y> (Type Y or N.)

Edit attributes of which blocks? <*>: (Press Enter for all, or specify the block names.)

Edit attributes with these names <*>: (Press Enter for all, or specify the attribute names.)

Edit attributes with this text <*>: (Press Enter for all, or specify the value names.)
Select attributes: (Select one or more attributes.)
Entities in set: 1
Select attributes: (Press Enter to end entity selection.)
1 attributes were selected.
Text to change: (Enter existing attribute value.)
New text: (Enter replacement value.)

Command Options

Option	Description
Edit attributes of which blocks	Specifies the names of blocks. You can use wildcards to specify similarly named blocks: • * refers to all characters in the block's name. • ? refers to individual characters.
Edit attributes with these names	Specifies the names of attributes. You can use wildcards to specify similarly named tags.
Edit attributes with this text	Specifies the values of attributes. You can use wildcards to specify similarly named values.
Select attributes	Chooses the attributes to edit.
Position	Moves the attribute's insertion point: Enter text insertion point - pick a new point, or enter coordinates.
Angle	Changes the rotation angle of the attribute text. Prompts you: New rotation angle <0> - enter a new angle, or pick two points to specify the angle.
Text	Changes the value of the attribute. Prompts you: [Change/Replace]<Replace> - type C or R. • **Text to change** searches and replaces text. • **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 styles.) New text style or ENTER for no change <Standard> - enter a different style name, or press Enter to keep the style.
Color	Changes the color of the attribute text: New color <BYLAYER> - enter a different color name or number.
Height	Change the height of the attribute text: New text height <0.2> - enter a different text height, pick two points in the drawing, or press Enter to keep the height.
Layer	Changes the layer of the attribute; the layer name must exist in the drawing. (Use the Layer command to create layers.) New layer or ENTER for no change <0> - enter a different layer name, or press Enter to keep the layer.
Next	Goes to the next attribute; if there are no more attributes to edit, exits the command.
pRev	Goes to the previous attribute, if any.

Edit only attributes visible on screen	Determines which group of attributes to edit: • Y (yes) edits only attributes visible in the current viewport. • N (no) edits all attributes in the drawing.
Text to change	Specifies the attribute value to change.

New text	Specifies the replacement value.
Quit	Exits the command.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
DdAtte - edits the values of attributes through a dialog box.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

AttExt

Copies data from attributes to a text file via a dialog box.

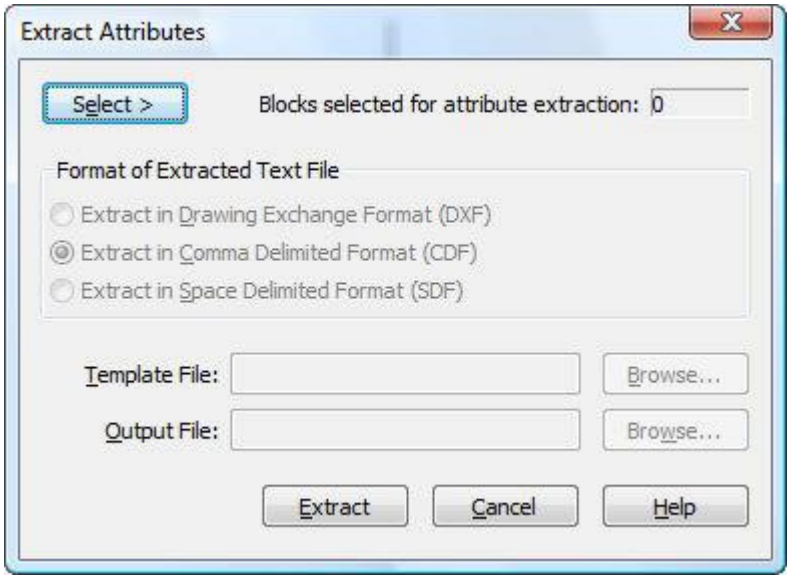
Deprecated. Use DataExtraction instead.

Accessing the Command

command bar: *attext*
aliases: *ax, ddattext*

: ***attext***

Displays a dialog box:



Enter options, and then click Extract.

If you select blocks without attributes and non-block entities, BricsCAD ignores them.

The template file determines which attributes are extracted and the format of related data. For example, a template.txt file contains the following fields:

BL:NAME C008000

BL:NUMBER N003000

PRODUCT C016000

BL:X N016004

BL:Y N016004

This means that the AttExt command will extract the following values:

- Name of the block formatted to 8 characters.
- Count of the block, formatted to a 3-digit number.
- Values of the attribute with the "Product" tab, formatted to 16 characters.
- The x and y coordinates of the block's insertion point, formatted to 16 digits, including 4 decimal places.

The table below lists all available data extraction parameters. The table uses the follow codes:

C = indicates alphanumeric field

N = indicates numeric field
fff = indicates width of field, in characters (max = 999)
ddd = indicates number of decimal places (max = 999)
0 = indicates zero.

Data	Format
Attribute tag (numeric)	Nfffddd
Attribute tag (alphanumeric)	Cfff000
Block name	BL:NAME Cfff000
Block counter	BL:NUMBER Nfff000
Block handle (identifier)	BL:HANDLE Cfff000
Block's layer name	BL:LAYER Cfff000
Block nesting level	BL:LEVEL Nfff000
Block insertion, x coordinate	BL:X Nfffddd
Block insertion, y coordinate	BL:Y Nfffddd
Block insertion, z coordinate	BL:Z Nfffddd
Block's rotation angle	BL:ORIENT Nfffddd
Block's x scale factor	BL:XSCALE Nfffddd
Block's y scale factor	BL:YSCALE Nfffddd
Block's z scale factor	BL:ZSCALE Nfffddd
Block's x extrusion direction	BL:XEXTRUDE Nfffddd
Block's y extrusion direction	BL:YEXTRUDE Nfffddd
Block's z extrusion direction	BL:ZEXTRUDE Nfffddd

Command Options

Option	Description
Select	Dismisses the dialog box temporarily so that you can select blocks with attributes: Select entities - choose one or more blocks with attributes. Select entities - press Enter to return to the dialog box.
Extract in Drawing Exchange Format (DXF)	Exports the attribute values in DXF format as .dxx files. This format is meant for use with other computer programs that can read .dxf files.
Extract in Comma Delimited Format (CDF)	Exports the attributes with commas separating values. This format is meant for use with spreadsheets and databases.
Extract in Space Delimited Format (SDF)	Exports the attributes in fixed-width format. This format is meant for use in word processors.
Template File	Specifies the name of the .txt template file. The file must exist before this command is used.
Browse	Displays the Attribute Extract Template File dialog box. Choose a file, and then click Open.
Output File	Specifies the name of the output file,
Browse	Displays the Attribute Extract Template File dialog box. Specify a file name, and then click Save.
Extract	Exports the attribute values to the output file.

Procedures

Extracting attribute information

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

-AttExt

Copies data from attributes to a text file via the command line.

Accessing the Command

command bar: *-attext*
alias: *-at*

: ***-attext***

Select Entities or **extract attibutes as** [**Entities**/Dxf/Cdf/Sdf] <CDF>: (Enter an option or press Enter.)

BricsCAD displays the Attribute Extract Template File dialog box.

Choose a .txt file, and then click Open.

The template .txt file must exist before this command is used. See the **AttExt** command for the format of this file.

BricsCAD displays the Attribute Extract Template File dialog box. Specify a file name, and then click Save.

The attribute values and other data are extracted, and saved to the output file. BricsCAD then reports the number extracted:

15 records extracted.

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

Command Options

Option	Description
Select Entities	Prompts you to select entities. Enter E for the following prompts: Select entities - choose one or more blocks with attributes. Select entities - press Enter to continue the command. If you ignore this option by entering DXF, SDF, or CDF directly, then BricsCAD selects all attributes in the drawing.
Extract attributes as: DXF/CDF/SDF	Specifies the data format: • DXF - exports the attribute values in a partial DXF format; this format is meant for use with other computer programs that can read .dxx files (DXf eXtraction). • CDF - exports the attributes with commas separating values; this format is meant for use with spreadsheets and databases. • SDF - exports the attributes in fixed-width format; this format is meant for use in word processors.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
AttExt - (deprecated) extracts attributes from drawings to data files through a dialog box.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

AttRedef

Redefines a block and updates associated attributes.

Accessing the Command

command bar: *attredef*

: ***attredef***
Prompts you in the command bar:

Enter name of block you wish to redefine: **(Type the name of an existing block.)**

Select entities for block: **(Select entities and attributes to be included in the block definition.)**

Entities in set: xx

Select entities for block: **(Select more entities or press Enter to stop selecting entities.)**

Specify new base point: **(specify the origin point of the block.)**

The following rules apply to existing block references:

- New attributes use default values.
- Values of old attributes that are included in the new block definition are preserved.
- Old attributes that are not included in the new block definition are deleted.
- Format or property changes made with the ATTEDIT or EATTEDIT commands are removed.
- Extended data associated with the block are deleted.

Command Options

No options.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines how a block's attributes will be created at the command prompt.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.

AttSync

Synchronizes attribute definitions in all block references of a specified block definition.

Accessing the Command

command bar: *attsync*

: ***attsync***
Prompts you in the command bar:
Enter an option [Select/Name/?]<Select>: (Press Enter or choose an option.)
Select a block: *(Select a block in the drawing.)*
ATTSYNC block <BlockName>? [Yes/No]<Yes>: (Press Enter to synchronize all instances of the selected block.)

Command Options

Option	Description
Select	Prompts you to select a block in the drawing.
Name	Prompts you: **Enter name of block to sync or ? to list attributed blocks:** Type a name of an attribute block in the command bar or press Enter to display a list blocks that contain attributes in the current drawing.
?	Lists all blocks that contain attributes in the current drawing.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines how a block's attributes will be created at the command prompt.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.

Audit

Repairs drawings that are open.

Accessing the Command

command bar: *audit*
transparent: *'audit*
menu bar: *File | Drawing Utilities | Audit*

: ***comandName***
Prompts you in the command bar:
Fix any errors detected? [Yes/No] <No>: *(Type Y or N.)*

Typical response:

55 objects audited

Total errors found during audit 0, fixed 0

Command Options

Option	Description
Yes	Fixes errors, if BricsCAD finds any.
No	Searches for errors in the drawing database, but does not repair them.

Procedures

To analyze the integrity of the current drawing
To repair a drawing

Related Commands

ProxyInfo - displays a information about proxy entities, contained in the current drawing, in a dialog box.
Recover - recovers damaged drawings.
RecoverAll - recovers a drawing with all nested XRef drawings, each drawing is saved with .BAK copy and .ADT report.

AutoComplete

Sets the command line autocomplete mode.

Accessing the Command

command bar: *autocomplete*

: ***autocomplete***
Prompts you in the command bar:
Enter an automatic completion option [Append/List/System variables/Delay/ON/OFf] <ON>: *(Choose an option.)*
The autocomplete options are saved in the AUTOCOMPLETEMODE system variable.

Command Options

Option	Description
Append	Prompts you: Automatically append suggestions as you type? [Yes/No] <Yes>: • Yes: **automatically completes the command as you type.** • No: **does not automatically complete the command.**
List	Prompts you: Display a list of suggestions? [Yes/No] <Yes>: • Yes: **display a list of suggestions.** • No: **do not display a list of suggestions.**
System variables	Prompts you: Include system variables in suggestions? [Yes/No] <Yes>: • Yes: **Includes system variables in the list of suggestions. The suggestion list includes all commands and/or system variables and/or user preferences that contain the current character string.** • No: **Does not list system variables in the list of suggestions.**
Delay	Specifies the delay time before the list of suggestions displays. Prompts you: Specify the delay time in seconds <0.50>: **Type the desired delay time, then press Enter.**
ON	Switches autocomplete on.
OFF	Switches autocomplete off.

Procedures

Using autocomplete in the command bar

Related Commands

All commands.

B

Background

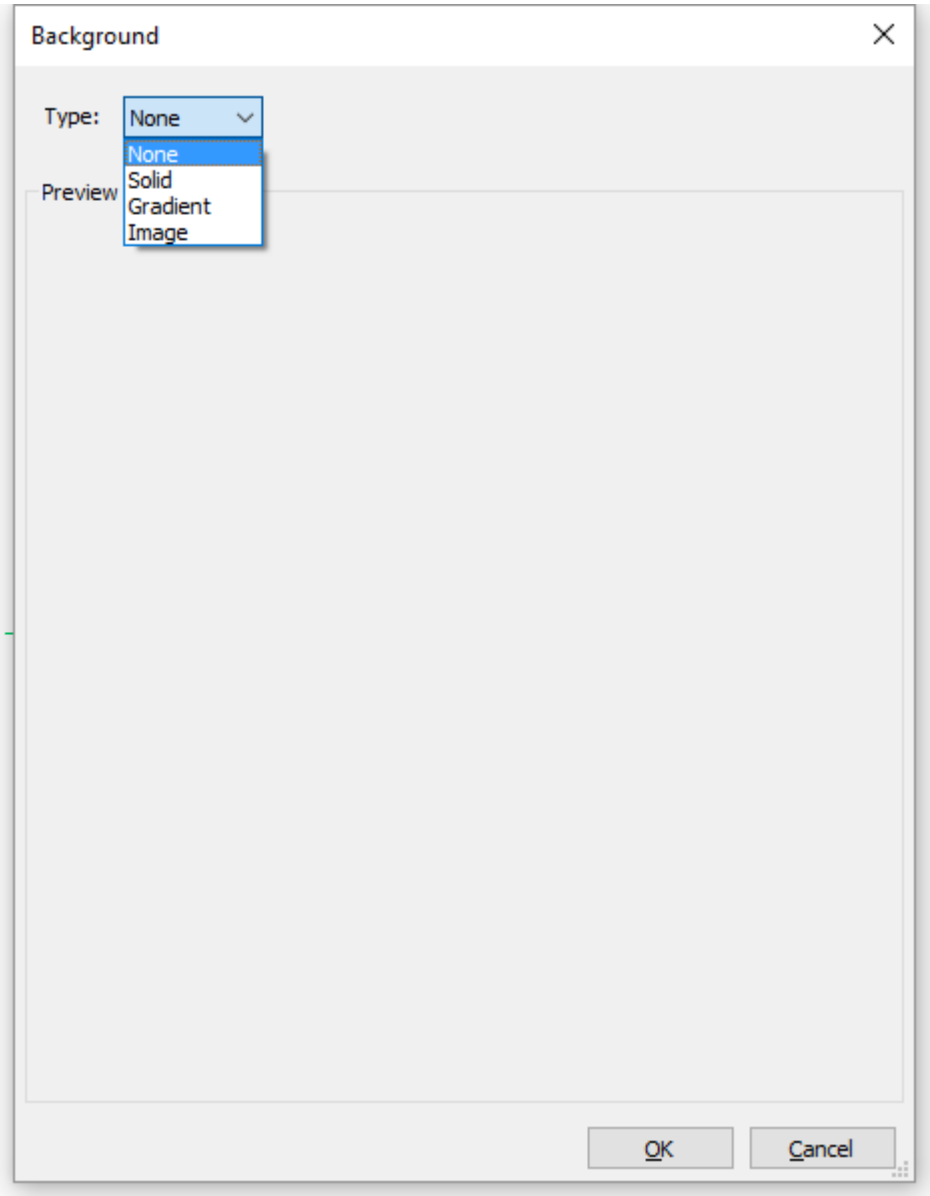
Sets the model space background for the current viewport and saved views.

Accessing the Command

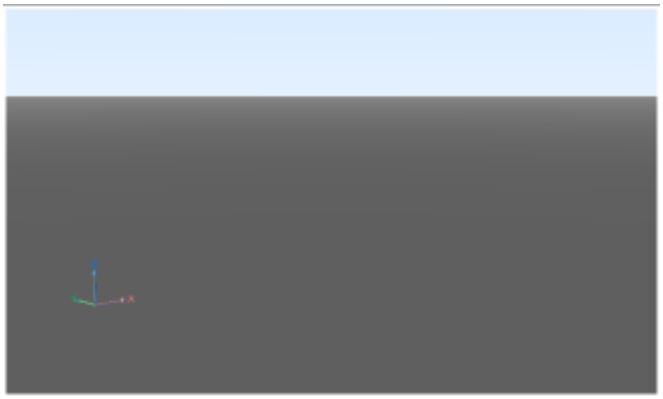
command bar: *background*

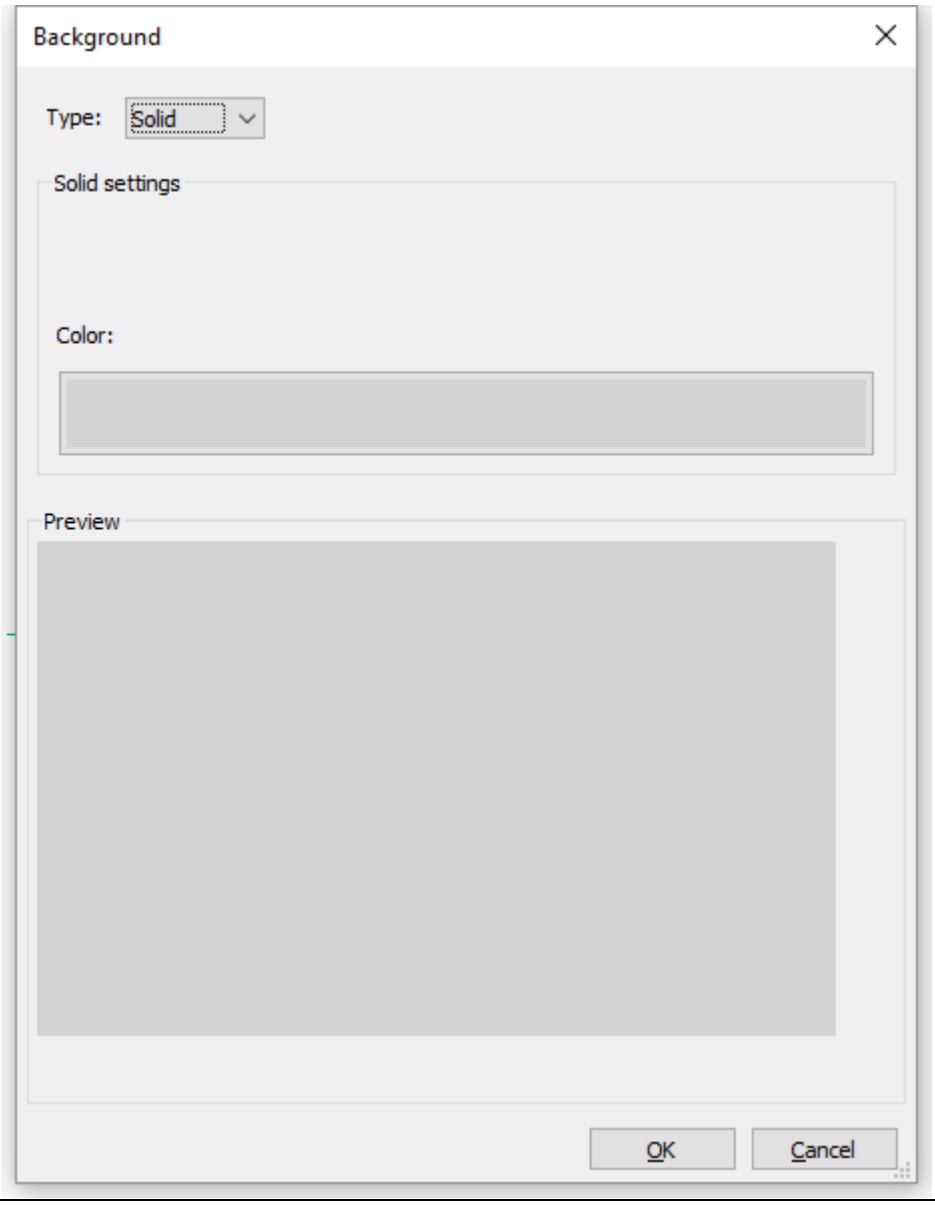
: ***background***

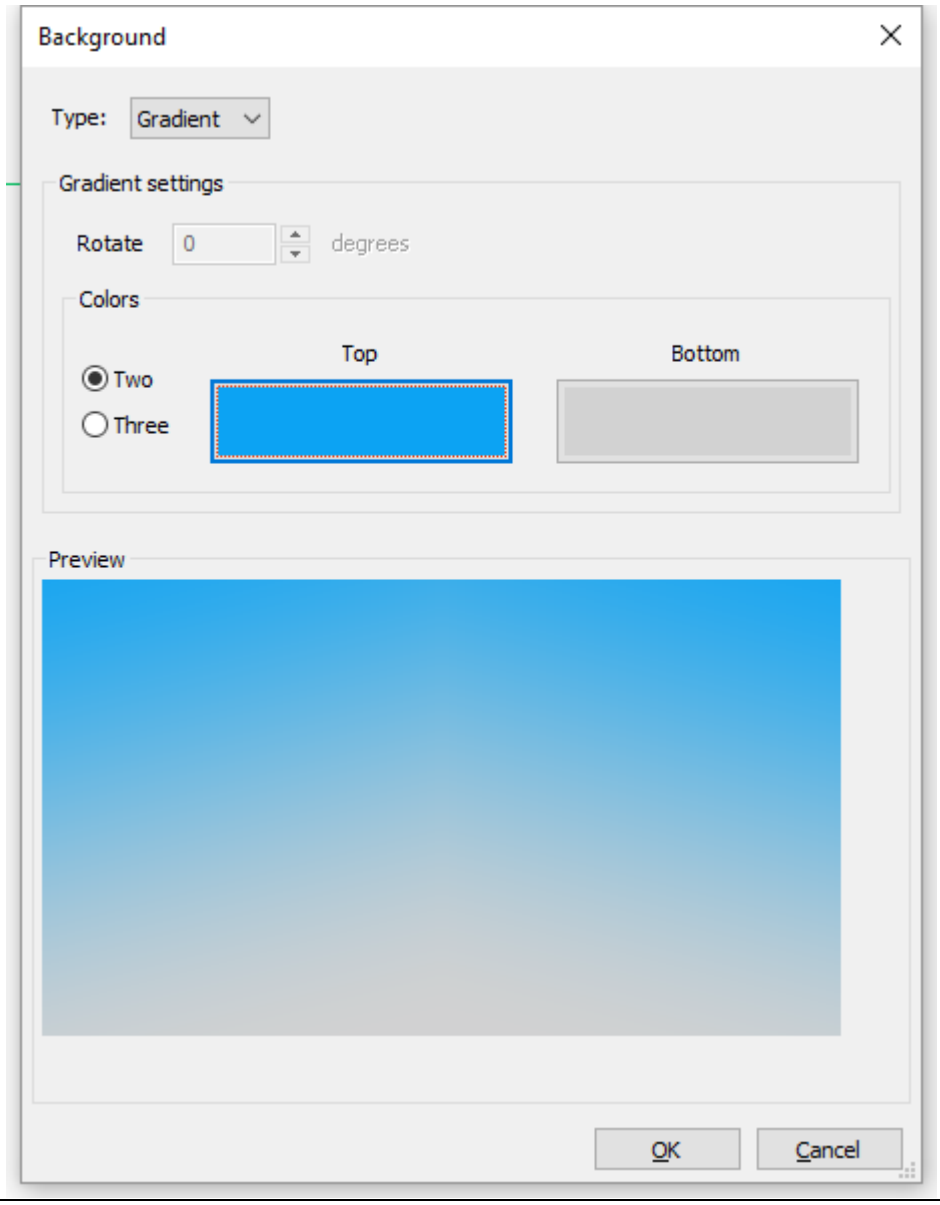
Displays a dialog box:



Command Options

Option	Description
None	Uses the default background. PERSPECTIVE = Off: white PERSPECTIVE = ON: horizon 
Solid	Sets the background to a solid color. Click the Color field to open the Color dialog box and select either an index color or a true color for the background.

	
Gradient	Sets the background to a gradient color. Select the gradient type: Two-color or Three-color. Click the *Top*, *Bottom* and *Middle* fields to open the Color dialog box and select either an index color or a true color.


Background
Type:
Gradient
Gradient settings
Rotate
0
degrees
Colors
Top
Bottom
Two
Three
Preview
OK
Cancel

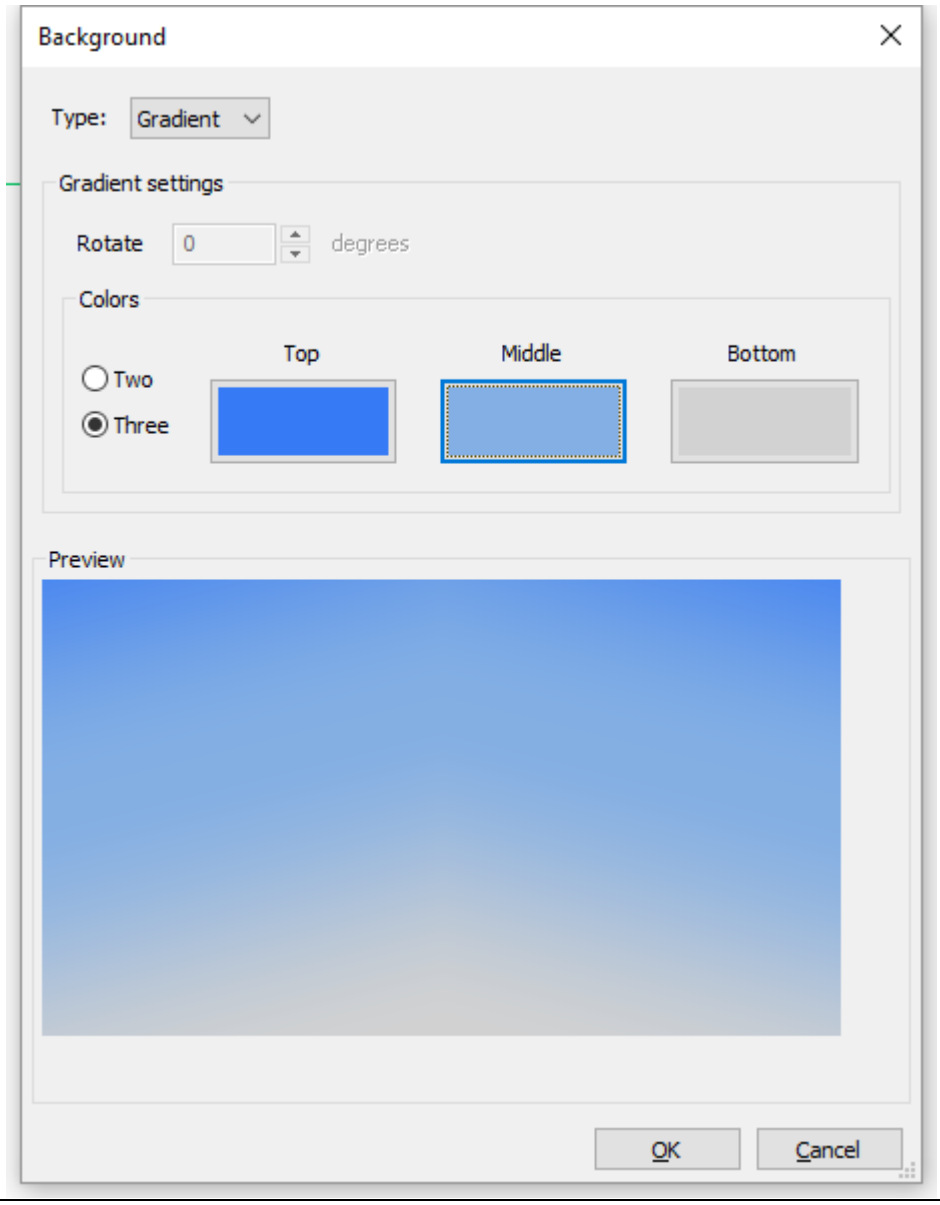
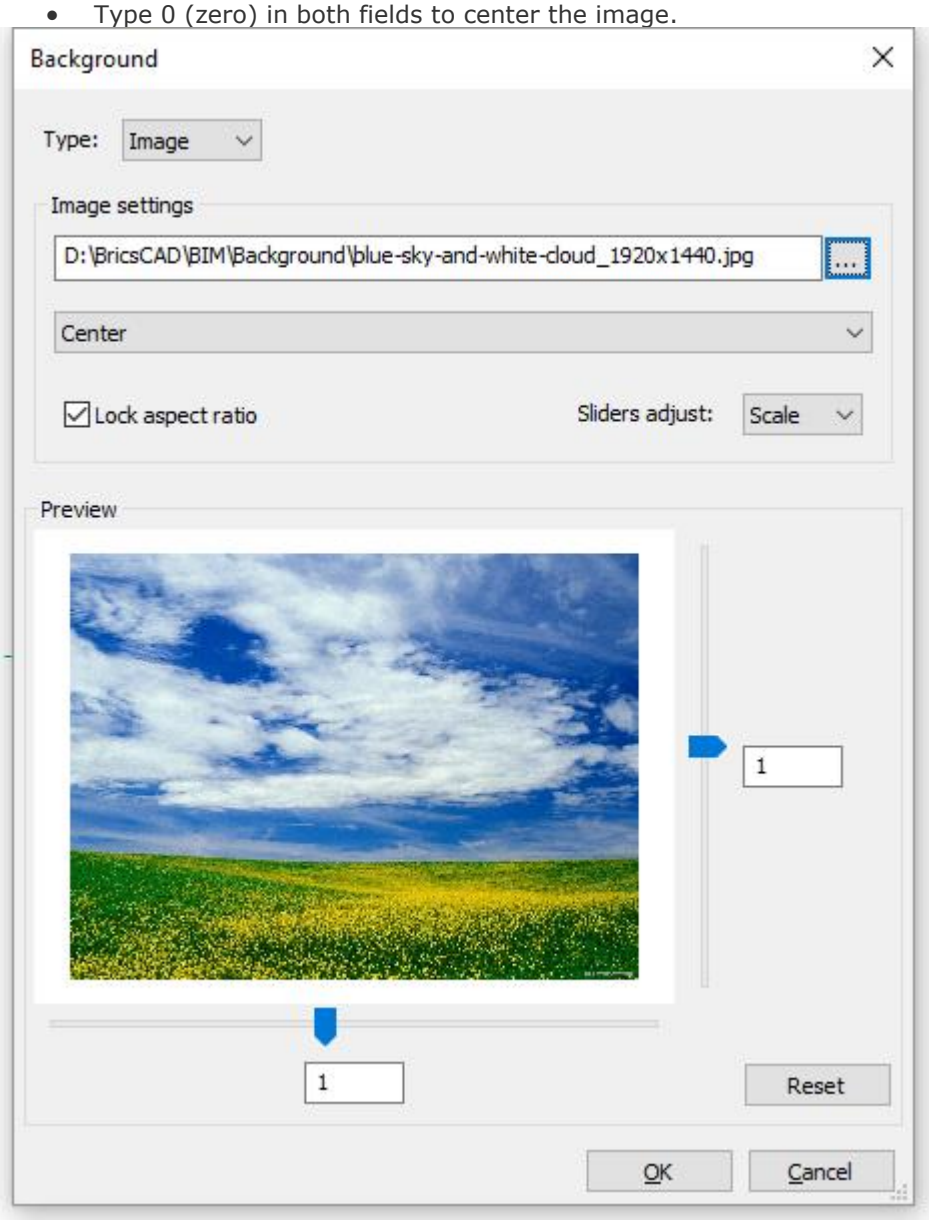
	

Image	Sets the background to an image.
	Click the *Browse* button (...) to select an image. Bmp, Gif, Jpg, Jpeg and Png image files are accepted.
	Optionally adjust the size and position of the image:
	• Select *Scale* from the *Sliders adjust* options.
	• Optionally check the *Lock aspect ratio* option to preserve the width/height ratio of the image.
	• Do one of the following to adjust the size of the image:
	  • Use the sliders at the right and bottom of the *Preview* pane.
	  • Type a value in the width (bottom) and Height (right) scale fields.
	• Optionally, uncheck the *Lock aspect ratio* option, then make the width scale factor negative to mirror the image.
	• Uncheck the *Lock aspect ratio* option, then elect *Offset* from the *Sliders adjust* options.
	• Do one of the following to move the image up/down:

- Drag the right-hand slider.
- Type a value in the right-hand field. Positive values move the image down.

- Do one of the following to move the image left/right.
  - Drag the bottom slider.
  - Type a value in the bottom field. Positive values move the image to the right.
  - Type 0 (zero) in both fields to center the image.



Related Commands

GradientBkgOff - switches the gradient background color of the current viewport off. Restores the default view background.
GradientBkgOn - restores the default gradient background of the current viewport.
MatchPerspective - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs.

Render - generates a photorealistic rendering using materials and lights through a dialog box.
-Render - generates a photorealistic rendering using materials and lights through the command bar.
View - creates and sets named views through the Drawing Explorer.

Base

Specifies the drawing's insertion point when inserted into other drawings.

Accessing the Command

command bar: *base*
transparent: *'base*
alias: *ba*
menu bar: *Settings | Base Point*
toolbar: *Settings |*

: ***base***
Prompts you in the command bar:
Base point for inserting this drawing <0.0000,0.0000,0.0000>: *(Enter x,y,z coordinates.)*

Command Options

Option	Description
Base point for inserting this drawing	Specifies x,y,z coordinates. The coordinates of the base point are in the current UCS. When this drawing is inserted into another drawing, these coordinates specify the insertion point, whether this drawing is inserted as a block or as an external reference.

Related Commands

-Insert - inserts DWG drawing files into the current drawing through the command bar.
Insert - inserts DWG drawing files into the current drawing through a dialog box.
WBlock - writes all or part of the drawing to a DWG file.
XRef - inserts DWG files as external references.

Battman

Manages the attributes of a block definition. Short for Block Attribute Manager.

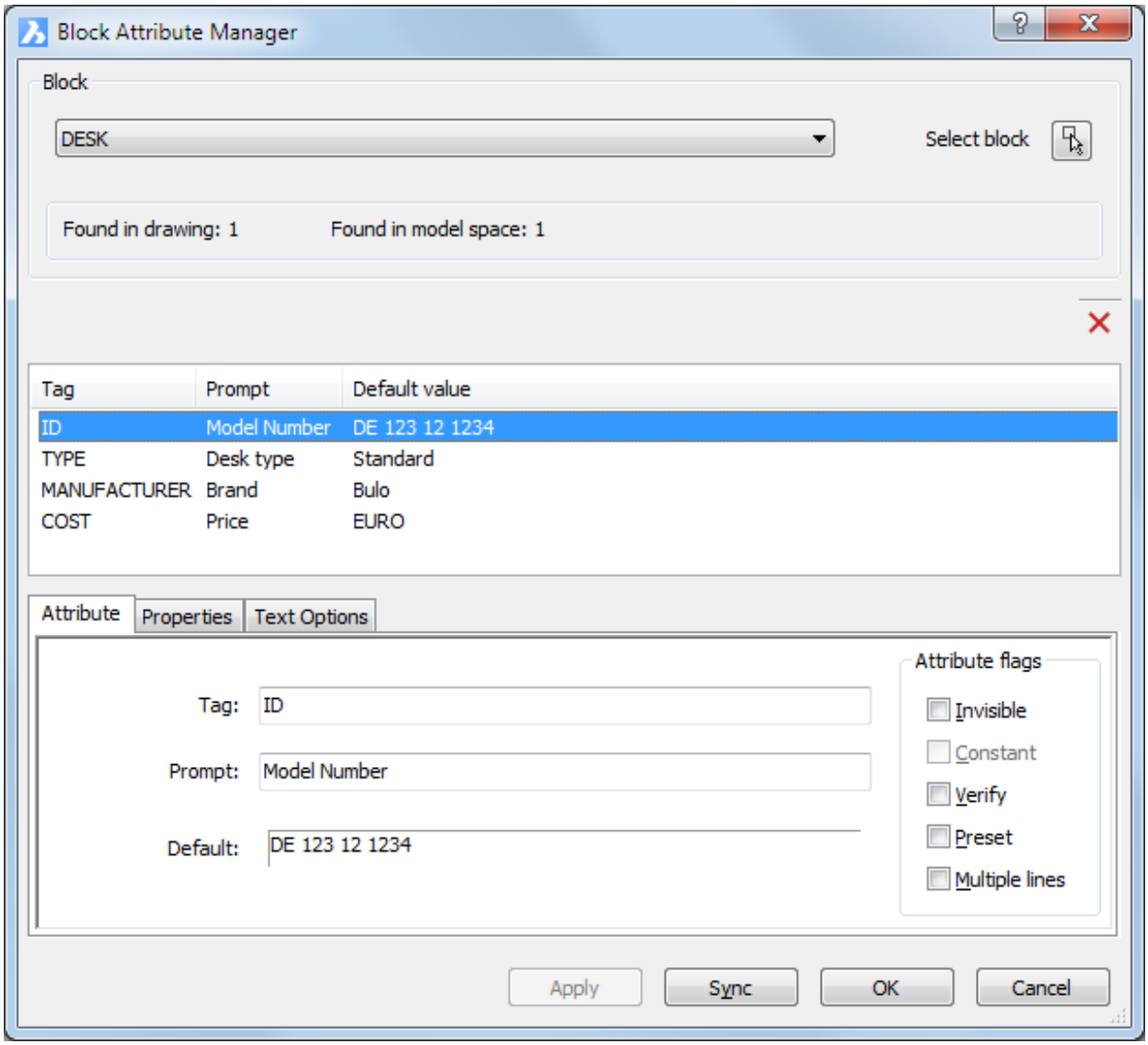
Accessing the Command

command bar: *battman*
menu bar: *Tools | Attributes | Block Attributes Manager*
toolbar: *Attributes |*

: ***battman***

Displays a dialog box.

If the current drawing does not contain attributes a message displays in the command bar.



Command Options

Option	Description
Block	Do one of the following: • Click the *Block* list button, then select a block in the list. • Click the *Select Block* button ( ). The *Block Attribute Manager* dialog box temporarily closes to let you select a block in the drawing.
	Select an attribute in the *Attribute* list. Tag / Prompt / Default value Type / Desk type / Standard Manufacturer / Manufactured by / Bulo ID / Model Number / DE 123 12 1234 Cost / Price? / €
✕	Deletes the selected attribute.
Attribute	Click the *Attribute* tab. The definition of the selected attribute displays. Tag: ID Prompt: Model Number Default: DE 123 12 1234 Attribute flags: Invisible, Constant, Verify, Preset, Multiple lines Click a field to edit or check/uncheck an attribute flag.
Properties	Click the *Properties* tab. The properties of the selected attribute display. Layer: 0 Color: ByLayer Lineweight: ByLayer Linetype: ByLayer Plot style: ByLayer
Text Options	Click the *Text Options* tab. The text settings of the selected attribute display. 2 Text Style: Arial Justification: Left Upside down Backwards Height: 0 Width Factor: 0 Rotation: 0 Oblique Angle: 0
Apply	Applies the changes to the block definition

Sync	Applies the changes to all block instances in the drawing.
OK	Saves all changes and closes the *Block Attribute Manager* dialog box.
Cancel	Undoes all changes and closes the *Block Attribute Manager* dialog box.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines how a block's attributes will be created at the command prompt.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.

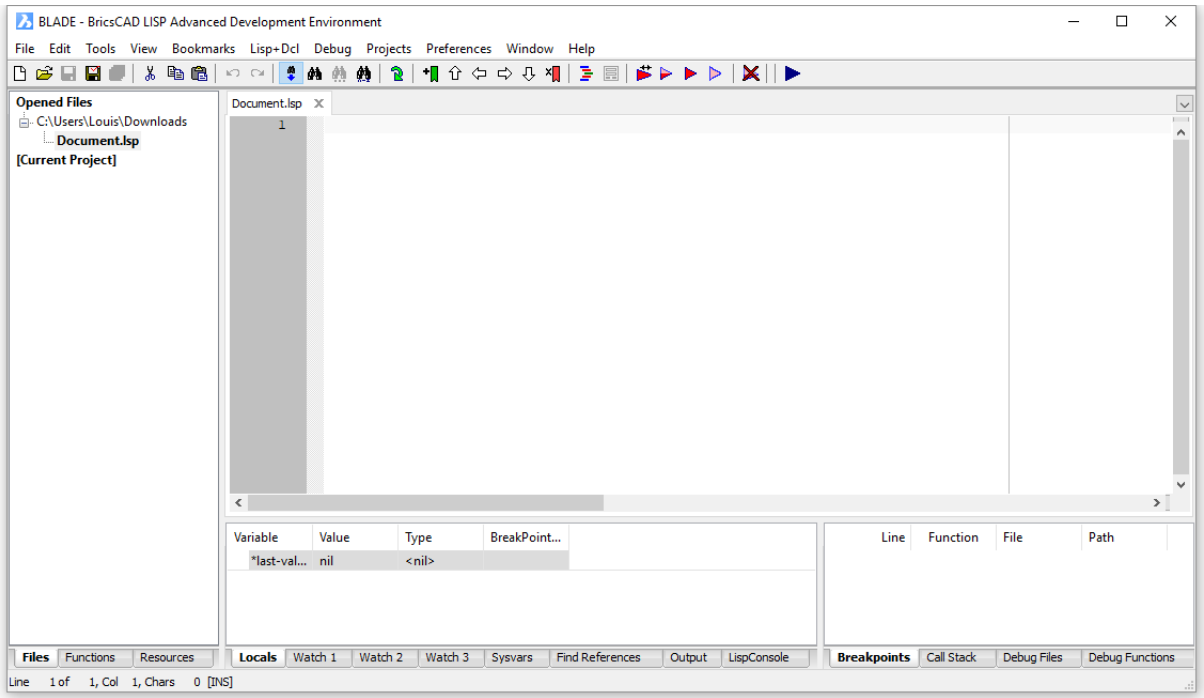
BLADE

Opens the advanced environment for editing and debugging LISP applications. Short for BricsCAD LISP Advanced Development Environment.

Accessing the Command

command bar: *blade*

: ***blade***
Displays BLADE - BricsCAD LISP Advanced Development Environmentdialog box:



Command Options

No command options.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
VbaLoad loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

BHatch

Fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box.

Accessing the Command

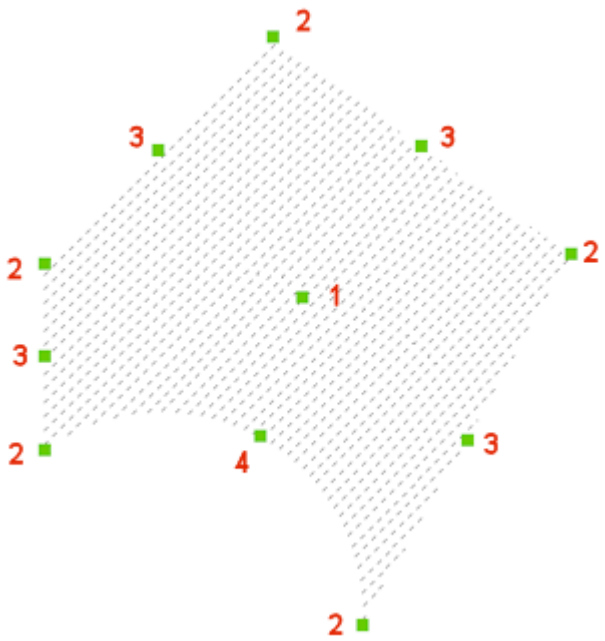
command bar: *BHatch*
aliases: *ba, h, hatch*
menu bar: *Draw | Hatch*
toolbar: *Draw 2D |*

:bhatch
See the Hatch and Gradient commands.

Grips Editing

Hatches can be edited directly through grips:

1. Select the hatch.
   Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.



1. Center grip: moves the hatch
2. Vertex grip: moves the vertex
3. Midpoint grip: moves the boundary segment
4. Arc midpoint grip: modifies the arc radius

Related Commands

Boundary and -Boundary place a polyline boundary inside closed areas.
Hatch - place hatch patterns through a dialog box.
-BHatch and -Hatch - places hatch patterns through the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Properties - changes the properties of hatch patterns.
Settings - specifies the default parameters for hatch patterns.
Solid - draws 3- and 4-side solid-filled areas.

-BHatch

Fills closed areas with repeating patterns; displays prompts at the command bar.

Accessing the Command

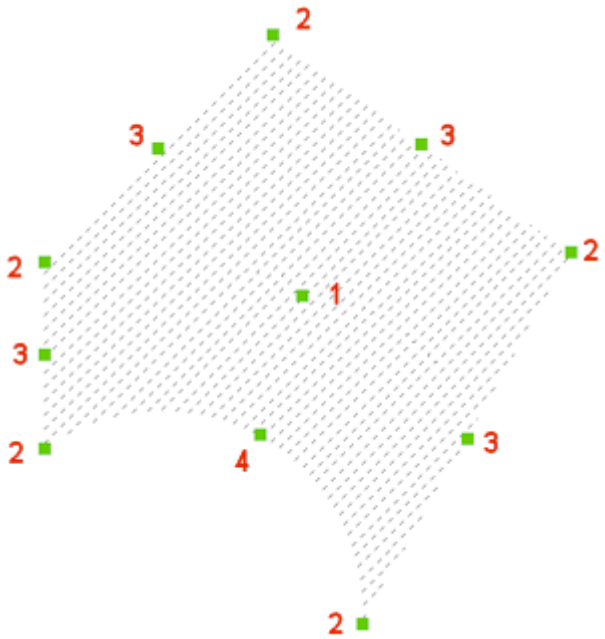
command bar: *-bhatch*
aliases: *-bh, h, -hatch*

: ***-bhatch***
See the -Hatch command.

Grips Editing

Hatches can be edited directly through grips:

1. Select the hatch.
   Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.



1. Center grip: moves the hatch
2. Vertex grip: moves the vertex
3. Midpoint grip: moves the boundary segment
4. Arc midpoint grip: modifies the arc radius

Related Commands

Boundary and -Boundary place a polyline boundary inside closed areas.
BHatch and Hatch - place hatch patterns through a dialog box.
-Hatch - places hatch patterns through the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.

Properties - changes the properties of hatch patterns.
Settings - specifies the default parameters for hatch patterns.
Solid - draws 3- and 4-side solid-filled areas.

Blipmode

Toggles the display of blips made when you pick points in the drawing.

Accessing the Command

command bar: *blipmode*
transparent: *'blipmode*
alias: *bm*

: ***blipmode***
Prompts you in the command bar:
Blipmode is currently off: ON/Toggle/<Off>: *(Enter an option.)*

Examples of blips made when the user picks points in the drawing.

Command Options

Option	Description
ON	Turns on the display of blips. Use the **Redraw** command to erase blips from the screen.
Toggle	Turns on blip mode, if off; turns off, if on.
Off	Turns off the display of blips.

Related Commands

Redraw - removes blips from the screen.
Settings - specifies the default parameters for blipmode.

Block

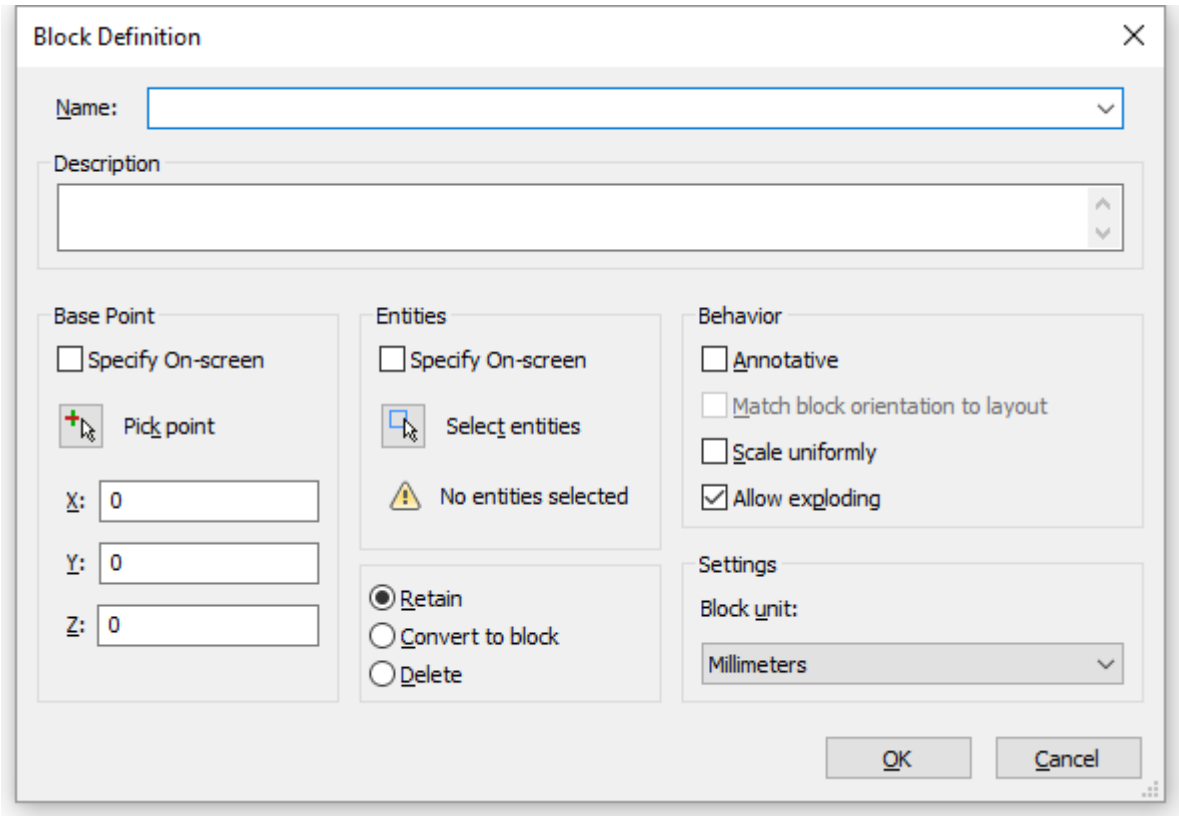
Groups entities into a block (symbol).

Accessing the Command

command bar: *block*
alias: *b*
menu bar: *Tools | Create Block*
toolbar: *Tools |*

: ***block***

Displays a dialog box:



Command Options

Option	Description
Name	Specifies the name of the block.
Description	(optional) Describes the block.
Base Point	Specifies the block's base point, the point at which it will be inserted into drawings with the Insert command. Specify on Screen: If checked, you are prompted to define the base point after clicking the OK button. Pick point: The *Block Definition* dialog box temporarily closes. You are prompted to specify the base point on screen. X Y Z: Type the coordinates of the base point in the x-, y- and z-fields.

Entities	Selects the entities that make up the block. Specify on Screen: If checked, you are prompted to select the entities after clicking the OK button. Select entities: The *Block Definition* dialog box temporarily closes. You are prompted to select the entities in the drawing Retain: The selected entities are retained in the drawing. Convert to block: The selected entities are converted to a block. Delete: The selected entities are deleted after the block definition is created.
Behavior	Annotative: Sets the *Annotative* property of the block. Match block orientation to layout: If checked, forces annotative blocks to display upright, regardless of the orientation of the viewport content. Scale uniformly: If checked, it is not possible to define the x-, y- and z-scaling of the block differently. Allow exploding: If checked, you can explode the block after placement. You can change this property in the Drawing Explorer - Blocks.
Settings	Block unit: Press the button, then select a unit from the list. The *Block unit* setting is used to scale the block correctly when the block definition is copied to another drawing of which the *Insertion Units* setting (INSUNITS) is different from the current drawing.
OK	Concludes the *Block* command. Saves the block definition in the drawing.
Cancel	Concludes the *Block* command without saving.

Procedures

General procedure to create a block
To create a block in a drawing
To create a block using the Blocks Explorer
To save a block to a separate file

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
CopyBase - copies entities to the Clipboard with a base point.
ExpBlocks - opens the Blocks section of the Drawing Explorer dialog
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
WBlock - writes blocks and other drawing parts to files on disk through a dialog box
-WBlock - writes blocks and other drawing parts to files on disk through a dialog box

-Block

Groups entities into a block (symbol).

Accessing the Command

command bar: *-Block*
aliases: *-b*

: ***-block***
Prompts you in the command bar:

Name for new block or [? to list existing blocks(?)]: **(Enter a name, or ?.)**

Insertion point for new block or [Annotative]: **(Pick a point, or enter x,y coordinates.)**

Select entities for block: **(Select one or more entities.)**

Select entities for block: **(Select more entities, or press Enter to exit command.)**

Command Options

Option	Description
Name for new block	Specifies the name of the block.
? to list existing blocks	Lists the names of blocks already in the drawing. Press F2 to display Text window.
Insertion point for new block	Specifies the block's base point, the point at which it will be inserted into drawings with the Insert command.
Select entities for block	Selects the entities that make up the block; you may use any selection method, including entity snaps. Press Enter to finish selecting entities and to exit the command. The entities disappear from view, but are now part of the block; use the **Oops** command to return the entities.

Procedures

General procedure to create a block
To create a block in a drawing
To create a block using the Blocks Explorer
To save a block to a separate file

Related Commands

CopyBase - copies entities to the Clipboard with a base point.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
WBlock - writes blocks and other drawing parts to files on disk through a dialog box
-WBlock - writes blocks and other drawing parts to files on disk through a dialog box

BmpOut

Exports the current viewport as a BMP (bitmap) file.

Accessing the Command

command bar: *bmpout*
menu bar: *File | Export*

: ***bmpout***
Displays a File dialog box.

Command Options

Option	Description
Save in	Selects the folder and drive in which to save the file.
File name	Specifies the name of the file.
Save	Saves the contents of the current viewport as a raster image.

Related Commands

Export - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats.
MSlide - exports drawings in WMF, EMF, and SLD formats.
WmfOut - exports drawings in WMF, EMF, and SLD formats.

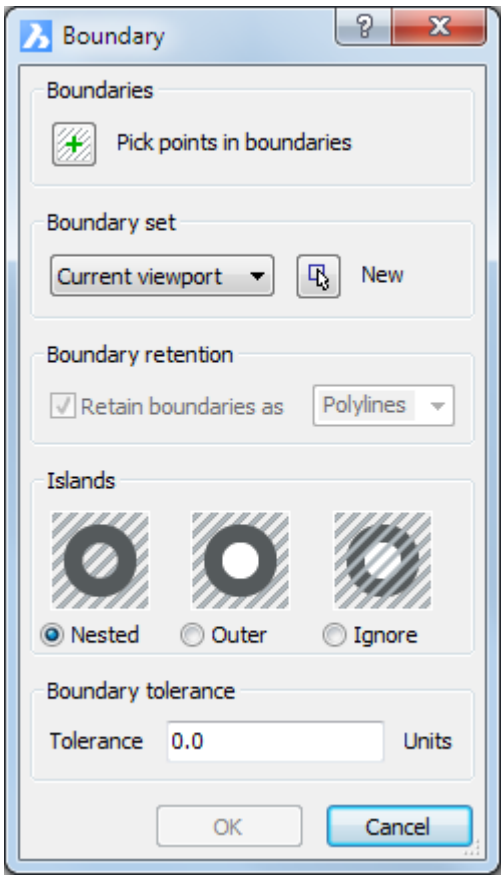
Boundary

Creates a polyline that forms a boundary inside a closed area through a dialog box.

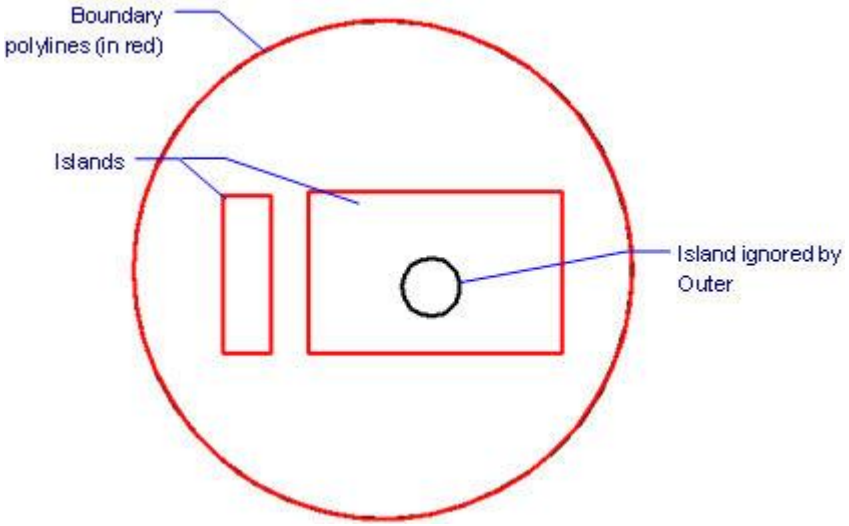
Accessing the Command

command bar: *boundary*
aliases: *bo, bpoly*
menu bar: *Draw | Boundaries | Boundary Polyline*
toolbar: *Draw | Boundaries|*

: ***boundary***
Displays a dialog box:



Choose options, and then click OK.



Command Options

Boundaries	Description
Pick points in boundary	Specifies the closed areas in which to place the boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select a point to define a boundary or hatch area - pick a point inside a closed area. Notice that BricsCAD highlights draws the boundary, allowing you to preview it. The prompt repeats, allowing you to choose additional areas. When done selecting areas, press Enter: Select a point to define a boundary or hatch area - press Enter to return to the dialog box.
Boundary Set	
Boundary set	Specifies where BricsCAD should search for entities that make up the boundary: • Current Viewport - searches all entities in the current viewport. • Existing Set - searches only the current selection set for possible boundaries.
New	Creates a new selection set of entities that make up the boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select entities - use any selection method to choose one or more entities. The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter: Select entities - press Enter to return to the dialog box.
Boundary Retention	
Retain Boundaries as (read only)	Polylines

Islands	
Nested	When an area contains other areas, BricsCAD creates multiple boundaries.
Outer	When an area contains other areas, BricsCAD creates a boundary of the outermost entities.
Ignore	When an area contains other areas, BricsCAD creates one boundary.
Boundary Tolerance	
Tolerance	Specifies the largest gap BricsCAD ignores when creating a boundary inside an area that is not fully closed.

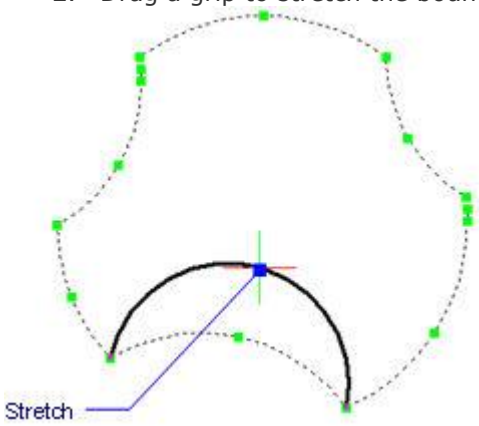
Procedures

Drawing polylines
Creating boundary polylines

Grips Editing

Boundaries can be edited directly through grips:

1. Select the boundary. Notice that it has many grips.
2. Drag a grip to stretch the boundary.



Related Commands

BHatch and Hatch - place boundaries and hatch patterns through a dialog box.
-BHatch and -Hatch - places boundaries and hatch patterns through the command bar.
-Boundary - creates a polyline boundary inside closed areas through the command bar.
GenerateBoundary - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Properties - changes the properties of boundaries.
Region - creates boundary-like regions.
Settings - specifies the default parameters for boundaries.
XEdges - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

-Boundary

Creates a polyline boundary inside closed areas through the command bar.

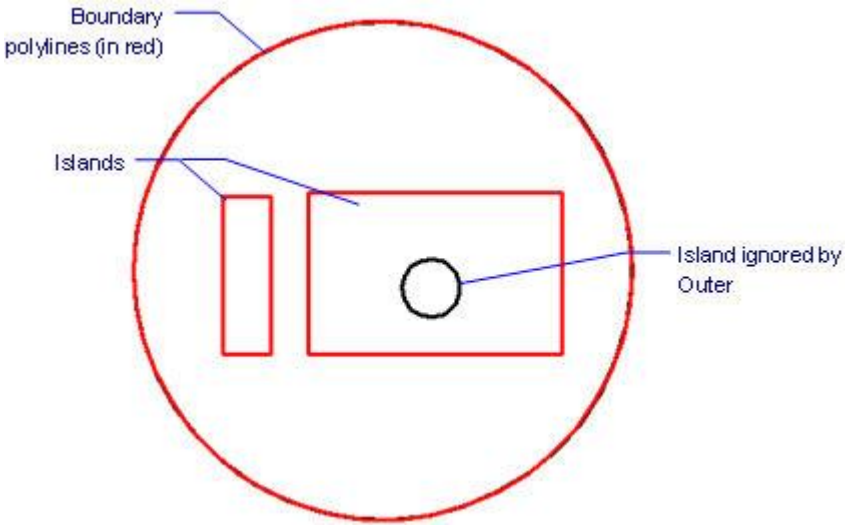
Accessing the Command

command bar: *-Boundary*
alias: *-bo*

: ***-boundary***
Prompts you in the command bar:

Advanced/<Internal point>: **(Pick a point inside a closed area, or type A.)**

Advanced/<Internal point>: **(Pick a point inside another closed area, enter A, or press Enter to end the command.)**



Command Options

Option	Description
Internal point	Specifies a point inside the closed area; enter x,y coordinates or pick a point in the drawing.
Advanced	
Boundary set	Specifies the entities to consider for creating boundaries: Specify candidate set for boundary - enter E or N. • Everything - selects all entities in the current viewport. • New - prompts you to select entities.
Island detection	Toggles whether islands are hatched over or not: Do you want island detection? - enter an option. • Yes - all islands are made into boundaries, as well as the outermost area. • No - islands are ignored; boundary is created of outermost area only. • Outer only - ignores innermost areas; boundary is created between outermost area and islands.
eXit	Returns to the original prompt.

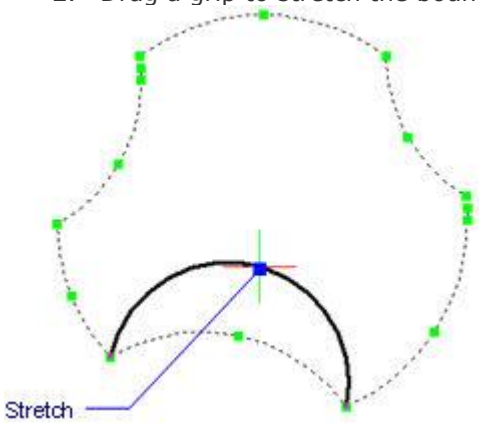
Procedures

Drawing polylines
Creating boundary polylines

Grips Editing

Boundaries can be edited directly through grips:

1. Select the boundary. Notice that it has many grips.
2. Drag a grip to stretch the boundary.



Related Commands

BHatch and Hatch - place boundaries and hatch patterns through a dialog box.
-BHatch and -Hatch - places boundaries and hatch patterns through the command bar.
Boundary places a polyline boundary inside closed areas through a dialog box.
GenerateBoundary - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Properties - changes the properties of boundaries.
Region - creates boundary-like regions.
Settings - specifies the default parameters for boundaries.
XEdges - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Box

Creates a box as a polyface mesh (Classic); Creates box as a 3D solid (Pro and Platinum).

Accessing the Command

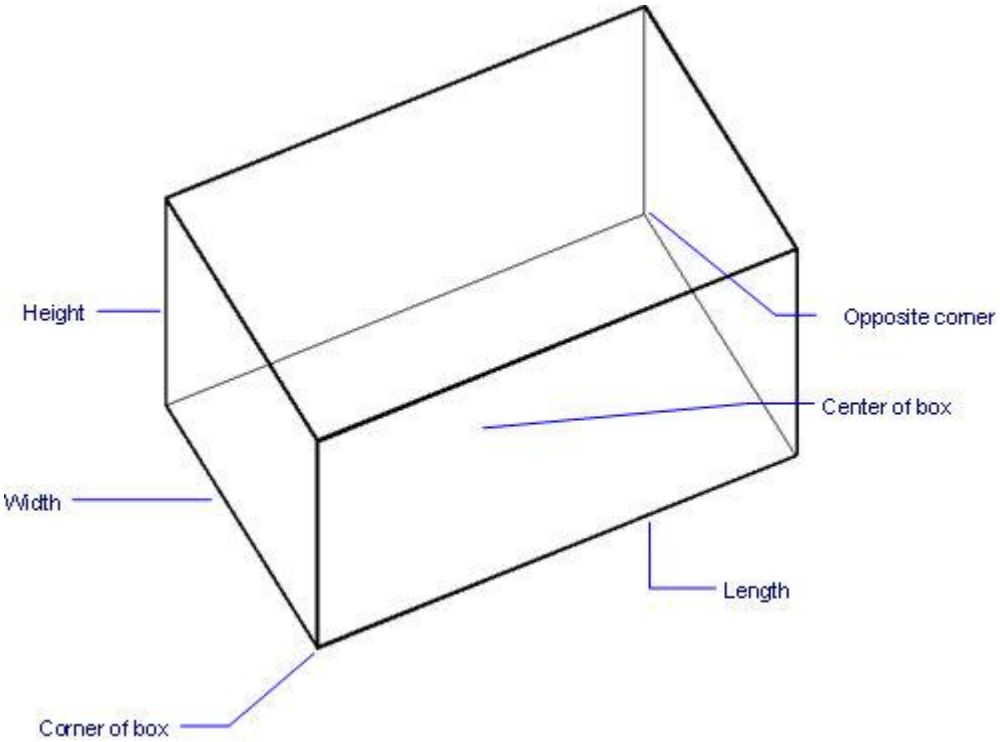
command bar: *box*
menu bar: *Model | 3D Solids | Box*
toolbar: *3D Solids |*

: ***box***
Prompts you in the command bar:

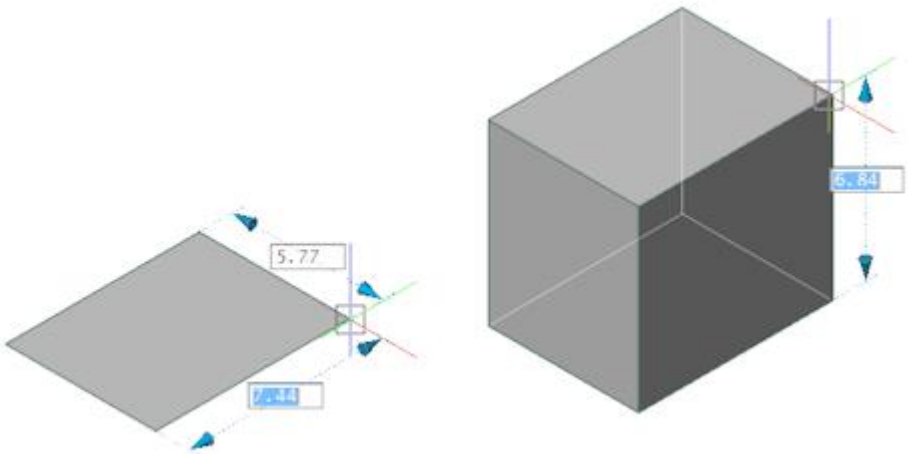
Set corner of box or [**Center**] <0.00,0.00,0.00>: (Pick a point, or enter an option.)

Set opposite corner or [**Cube**/**Length**)]: (Pick another point, or enter an option.)

Height of box [2Point]: (Pick two points, or enter a height.)



If Dynamic Dimensions are active, you can enter the dimensions in the dynamic dimensions entry fields.



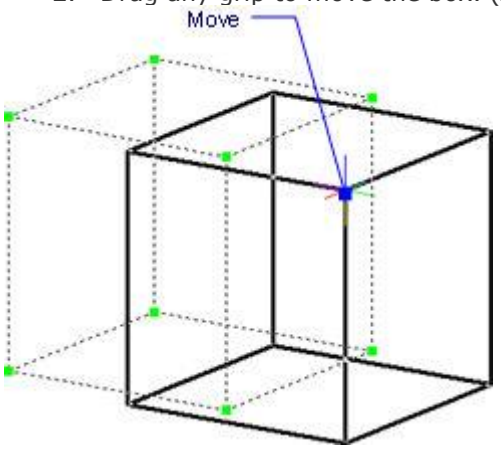
Command Options

Option	Description
Corner of box	Specifies one corner of the box's base. Enter x,y,z coordinates or pick a point in the drawing.
Center	Specifies the center of the box's base. Enter x,y,z coordinates or pick a point in the drawing.
Cube	Specifies a box with equal sides: Length of sides of cube - enter a length, or pick two points.
Length	Specifies the lengths of each side: Length of side of box - specify the length in the x direction. Width of box - specify the length in the y direction. Height of box - specify the length in the z direction. You may enter negative values to draw the box in the downward directions.
Opposite corner	Specifies the opposite corner of the box's base. Enter x,y,z coordinates or pick a point in the drawing.
Height of box	Specifies the height of the box. Enter a length or pick two points in the drawing.

Grips Editing

Boxes can be edited directly through grips:

1. Select the box. Notice that it has many grips.
2. Drag any grip to move the box. (All eight grips perform the same function: move.)



Related Commands

Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

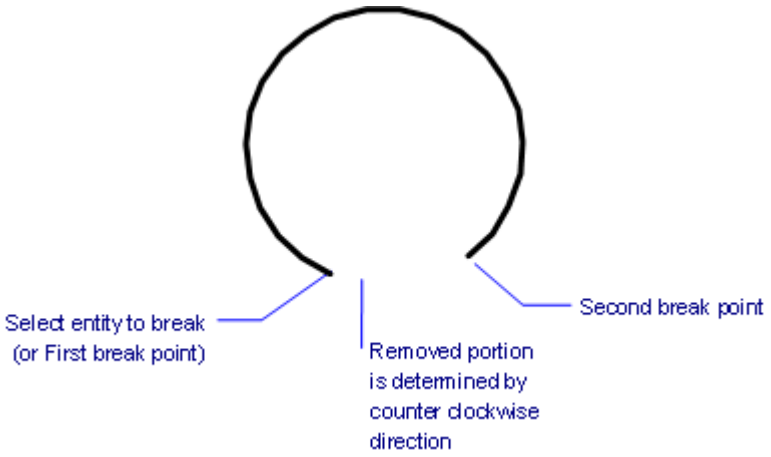
Break

Removes a portion of an entity.

Accessing the Command

command bar: *break*
alias: *br*
menu bar: *Modify | Break*
toolbar: *Modify |*

: ***break***
Prompts you in the command bar:
Select entity to break: *(Pick an entity.)*
Select second breakpoint or [First break point/@]: *(Pick a point on the entity or choose an option.)*



Command Options

Option	Description
Select entity to break	Selects the entity to break; only one entity can be selected. The pick point is also the first break point, unless you use the *First break point* option.
First break point	Specifies the start of the entity to remove.
Second break point	Specifies the end of the broken out section. On arcs and circles, the break occurs in the counter-clockwise direction.
@	Same point as first point. The selected entity is broken into two connected parts.

Procedures

To break an entity

Related Commands

DimBreak - breaks dimension lines, extension lines and leaders at locations where they cross other entities.

Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints..
PEdit - joins lines and arcs into a single polyline.
Trim - cuts entities where they cross other entities.

Browser

Opens the default Web browser.

Accessing the Command

command bar: *browser*
alias: *url*

: *browser*
Prompts you in the command bar:
Enter URL <http://bricsys.com>: *(Enter address of a Web page or press Enter to open the Bricsys website.)*

Command Options

Option	Description
Enter URL	Specifies a location on the Internet, such as www.bricsys.com. After you press Enter, your Web browser opens to that page. URL is short for "universal resource locator."

Related Commands

Url - opens the default Web browser.
OnWeb - opens the Bricsys home page in your computer's default Web browser.
Mail - attaches the current drawing to a message in the default email client.

BIM

BimAddEccentricity

Allows to control the relative position of the axis of a linear solid.

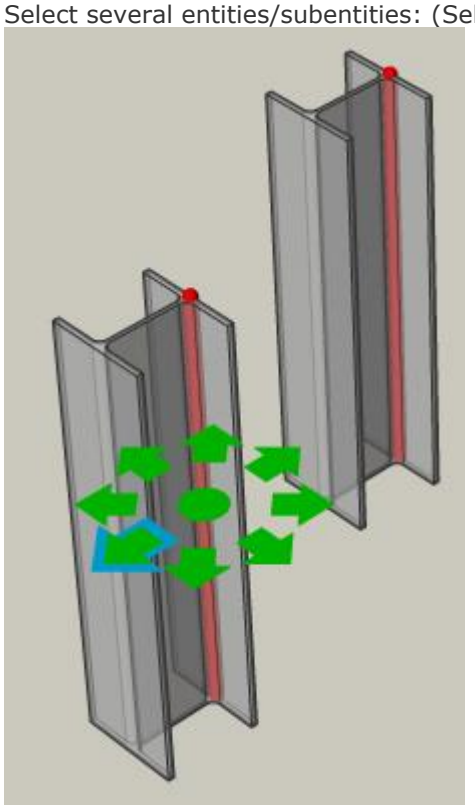
Accessing the Command

command bar: *bimaddeccentricity*
Quad: *Model*

: ***bimaddececcentricity***
Prompts you in the command bar:

Select several entities/subentities: (Select a linear solid.)
Entities in set: 1

Select several entities/subentities: (Select more linear solids or press Enter.)



Choose base location [Accept] <Accept>: (Click a new location; type a location or press Enter.)

Command Options

Option	Description
Click the widget	Click one of the nine locations, then right click to confirm.
Type a location	The options are: • TR: Top Right • TM: Top Middle • TL: Top Left • MR: Middle Right

	• C: Center (resets the axes of the selection set to the default location) • ML: Middle Left • BR: Bottom Right • BM: Bottom Middel • BL: Bottom right

Related Commands

BimApplyProfile - applies a profile to linear entities and linear solids.
BimLinearSolid - creates linear solids.
BimProfiles - opens the profiles library in the current project and the referenced library.
BimRecalculateAxis - recalculates the axis of linear building elements and repositions it back on the center line of a linear element.

BimApplyProfile

Applies a profile to linear entities and linear solids.

Accessing the Command

command bar: *bimapplyprofile*
menu bar: *BIM | Apply Profile*
toolbar: *BIM |*

: ***bimapplyprofile***
Prompts you in the command bar:

Select path(s): (Select a linear entity* or linear solid.)

Entities in set: 1

Select path(s): (Select more linear entities or linear solids or press Enter.)

Select profile [in Dialog] <in Dialog>: (Select a profile entity in the drawing or press Enter to display Profile dialog box.)

Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>: (Press Enter to apply the profile or choose an option.)

*Lines, polylines, arcs, circles, elliptical arcs and ellipses are accepted. Open and closed splines are accepted only if they have default tangents and are not self-intersecting.

Command Options

Option	Description
Select profile	Allows to select a profile in the model; prompts you: Pick a profile - Do one of the following: • Click a closed 2D entity or a region in the model. • Click an existing linear solid.
In dialog	Opens the Profiles dialog box. Select a profile, then click the OK button on the Profiles dialog box.
OK	Applies the profile to the selection.
Quarter Turn	Rotates the profile 90° counter clockwise.
Rotate	Allows to rotate the profile about a user defined angle; prompts you: Rotation angle <90> - type a value. Positive values rotate the profile counter clockwise.
Delete Clippings	Removes all clippings from the selected linear solids.

Related Commands

BimAddEccentricity - allows to control the relative position of the axis of a linear solid.
BimLinearSolid - creates linear solids.
BimProfiles - opens the profiles library in the current project and the referenced library.

BimAttachComposition

Attaches a BIM composition to solids.

Accessing the Command

command bar: *bimattachcomposition*
menu bar: *Bim | Attach Composition*
toolbar: *Bim |*

: ***bimattachcomposition***
Prompts you in the command bar:

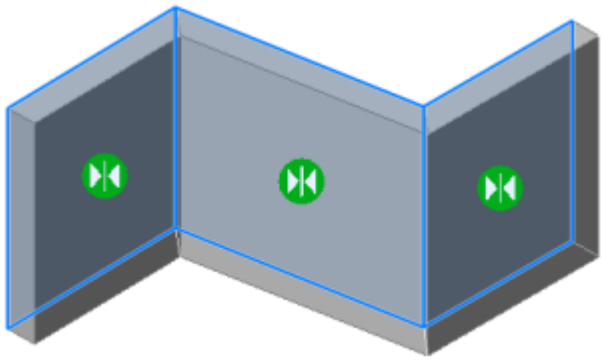
Enter composition name or [Dialog] <Dialog>: (Type a composition name or press Enter to display a dialog box.)

Select entities to attach composition: (Select entities in the drawing.)

Entities in set: 1

Select entities to attach composition: (Select more entities or press Enter to stop.)

The exterior face* of the selected solids highlights.



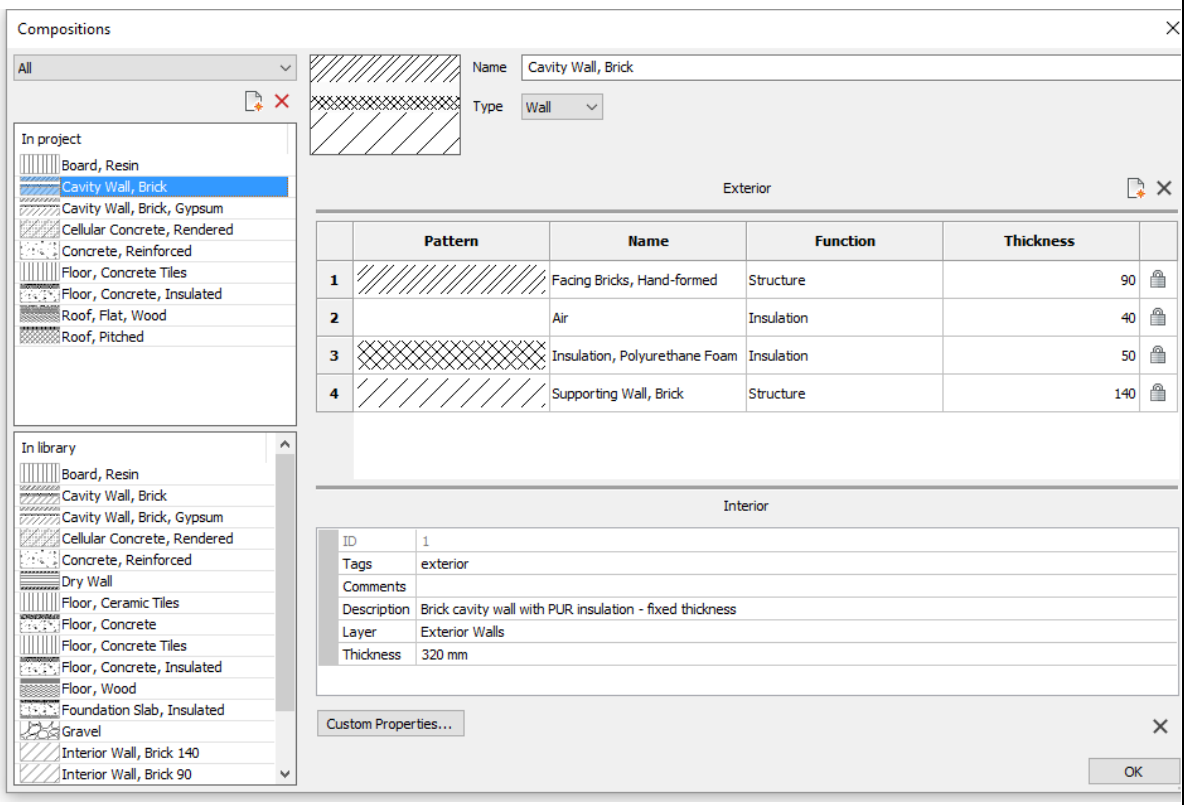
Choose exterior faces [Accept] <Accept>: (Press Enter to accept or click the *Flip Face* icon ( ) of one or more solids to flip the exterior face.)

The composition has been assigned to xx element(s).

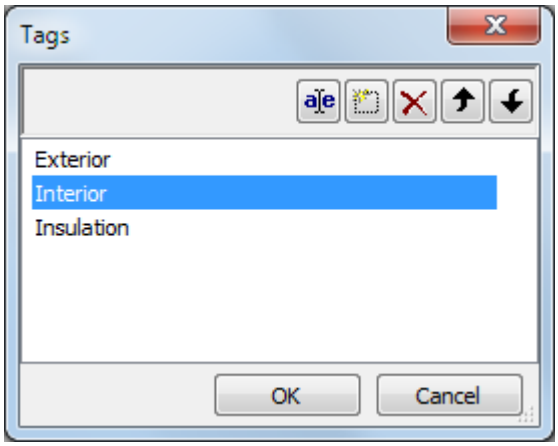
*For slabs the top face is the exterior face.

Command Options

Option	Description
Opens the *Compositions* dialog box:	



Select the composition type All	Click the button to select a composition type.: All Wall Slab Roof Generic • All: Lists all compositions • Wall: Lists compositions of the type *Wall* only. • Slab: Lists compositions of the type *Slab* only. • Roof: Lists compositions of the type *Roof* only. • Generic: List compositins of the type *Generic* only. The composition type is defined by the *Type* property of the composition. Although Compositions are grouped into *Wall*, *Slab*, *Roof* and *Generic* compositions, any type of composition can be attached to any dwg entity, including 2D entities.
In Project	Compositions which are already used in the Project database.
In Library	Compositions available in the Library database.
Structure grid	Material and thickness of the various composition layers. The layer on top is placed at the outside face of the solid. • Click the *New* button ( ) to add a new layer. • Select a layer, then click the *Delete* button ( ) to delete a layer. • Drag the number button in front of the pattern name to change the order of the layers. • Double click the *Pattern* or the *Name* field of a layer to edit the material. • Click the *Function* field to select a different function. By default the function is set in the *Material* definition. • Click the *Thickness* field to adjust the thickness of a layer. If the *Allow Custom Thickness* property of the material is *No*, the thickness cannot be modified. Double click the Thickness field to edit the material if necessary. • Click the padlock icon to unlock/lock a layer. Only one layer can have an unlocked thickness. If such layer exists, its thickness will be adjusted with respect to the thickness of the solid • : Locked thickness. When all layers have a locked thickness, the thickness of the solid will be adjusted to the overall thickness of the composition. The BIMUPDATETHICKNESS command re-applies the thickness of the composition to the solid in case the composition is modified.

	• : Unlocked thickness. The thickness of a layer can be unlocked only if the *Allow Custom Thickness* property of the material is *Yes*.
Properties grid	Properties of the selected material • ID: Non-editable ID number. • Type: Composition type. Click the field then select either *Wall*, *Slab*, *Roof* or *Generic*. • Tags: A series of user-defined tags. Tags are separated by commas. Tags are case sensitive. Select the field, then click the *Browse* button ( ) to edit the tags:  • Layer: sets the layer for the composition in the generated section. • Thickness: overall thickness of the composition. If one of the layers has an unlocked thickness, the minimum thickness of the composition displays.

Procedures

Attaching a composition to solids

Related Commands

Bimflip - flips the starting face from which the layers of a composition are set out.
BimProjectInfo - opens the BIM Project Info dialog box.
BimUpdateThickness - re-applies the overall thickness of a composition to the solid.
BLCompositions - opens the compositions library in the current project and the referenced library.
BLMaterials - opens the physical materials library in the current project and the referenced library.

BimAttachSpatialLocation

Allows to attach or remove a spatial location to a selection of entities.

Accessing the Command

command bar: *bimattachspatiallocation*
Quad: *BIM |*

: ***bimattachspatiallocation***
Prompts you in the command bar:*

Locations:

1. Building-2
2. Building-2 - Floor 0
3. Building-2 - Floor 1

...

* It might be necessary to press the F2 function key to open the Prompt History window.

Enter location number or [Auto attach locations/Unattach current location] <Auto attach locations>: (Enter a location number or choose an option.)

Select entities to update: (Select the entities you want to attach a spatial location to.)

Entities in set: xx

Select entities to update: (Select more entities or press Enter.)
The spatial location has been assigned to xx element(s).

Command Options

Option	Description
Auto attach locations	Allows to automatically assign a spatial location; prompts you: Select entities to update - select the entities to be processed. Entities in set: xx Select entities to update - select more entities or press Enter. Building 'Building': <Existed> The spatial location has been assigned to yy element(s). Skipped: zz element(s). If necessary, new building(s) and stories are created in the Building and Story Manager dialog box.
Unattach current locations	Allows to remove the current spatial location from a selection set; prompts you: Select entities to update - select the entities to be processed. Entities in set: xx Select entities to update - select more entities or press Enter. The spatial location has been removed from xx element(s).

Related Commands

BimClassify - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).
Bimify - analyzes the model and runs automatic classification and spatial location assignment for the entire model.
BimSpatialLocations - opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.

BimClassify

Classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).
Any dwg entity can be classified as a Building Element.

Accessing the Command

command bar: *bimclassify*
menu bar: *BIM | Classify As Building Element*
toolbar: *BIM |*
Quad: *BIM |*

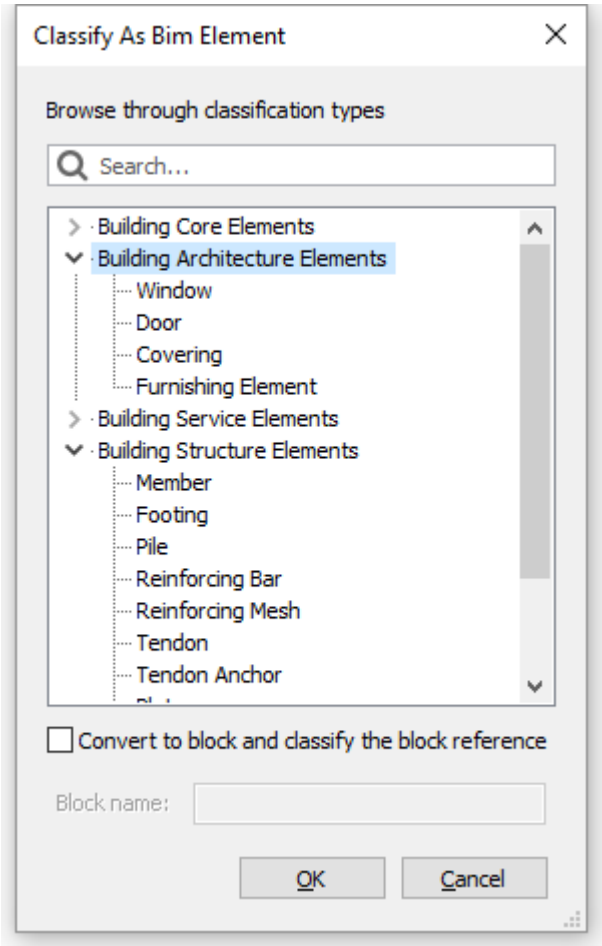
: ***bimclassify***
Prompts you in the command bar:

Classify entities as:
[Wall/Column/Slab/Beam/wIndow/Door/buildingElement/Other/Auto/Unclassify] <Other>: (Select an option.)

Select entities to classify <Drawing>: (Select ore or more entities or press Enter to classify the drawing as a building element.)

Command Options

Option	Icon	Description
Wall		Classifies the selection as wall elements. This option is applicable to any 3D Solid. Attaching a Wall composition to a solid automatically classifies the solid as a wall.
Column		Classifies the selection as column elements. This option is applicable to any 3D Solid.
Slab		Classifies the selection as slab elements. This option is applicable to any 3D Solid. Attaching a Slab or Roof composition to a solid automatically classifies the solid as a slab.
Beam		Classifies the selection as beam elements. This option is applicable to any 3D Solid.
Window		Classifies the selection as windows. This option is applicable to any block reference
Door		Classifies the selection as doors. This option is applicable to any block reference.
Building Element		Classifies the selection as unspecified building elements.
Other		Allows to classify the selection as building elements or convert the selection to a block and classify the block reference as a building element. Displays a dialog box:

		 Do the following: • (option) Type a search string in the *Search* field to filter the available element types. • Select a building element type. • (option) Check the *Convert to block ...* option and type a name in the *Block name* field. • Click the *OK* button.
Auto		Automatically classifies the selected entity or entities. Section entities become BimSection entities.
Unclassify		Removes all BIM data from the selected entities.
Drawing		Classifies the entire drawing as a building element, e.g. a window, a door or a curtain wall. When the entire drawing is classified a number of appropriate properties can be defined in the Properties Bar. The following properties are available for windows:

BIM	
Type	Window
Name	W_3x1_L
Building	
Story	
Composition	none
Display composition	Off
GUID	{D7FB5026-4C
Construction type	ALUMINIUM
Operation type	SINGLE_PANEL
Window insert	
Overall height	1200 mm
Overall width	1500 mm
Sill height	800 mm
Head height	0 mm
Assembly code	
Assembly subtype	

IFC Glazing	
Glass layers	2
Glass color	
Tempered	Off
Laminated	Off
Coated	Off
Wired	Off
Fill gas	
Translucency	1
Reflectivity	0
Beam radiation transmittance	0
Solar heat gain transmittance	0
Thermal transmittance summer	0
Thermal transmittance winter	0

IFC Shading	
External shading coefficient	0
Internal shading coefficient	0
Inset shading coefficient	0
Manufacturer	
Manufacturer	
Manufacturer URL	
Manufacturer product name	
Manufacturer product No	
Cost	0
Construction Specifiers Institute	

IFC Common	
Reference	
Acoustic rating	
Fire rating	
Thermal transmittance	0
External	On
Smoke stop	Off
Security rating	
Infiltration	0
Glazing area fraction	0

Definitions

Type	Description
	Building Core Elements
Wall	Vertical construction that subdivides spaces. Example: separation wall, bearing wall, interior wall
Beam	Structural member that carries loads between or beyond points of support, usually narrow in relation to its length and (nearly) horizontal. Example: I-beam, L-beam, T-beam, double T-beam
Column	Structural member of slender form, usually vertical, that transmits to its base the forces, primarily in compression, that are applied to it. Example: Doric column, Ionic column, Corinthian column
Slab	Component of the construction that normally encloses a space vertically. The slab may provide the lower support (floor) or upper construction (roof slab) in any space in a building Example: floor slab, roof slab
Stair	Construction comprising a succession of horizontal stages (steps or landings) that make it possible to pass on foot to other levels. Example: stairs, escalator, spiral staircases interrupted with landings
Stair Flight	Assembly of building components in a single "run" of stair steps (not interrupted by a landing). The stair steps and any stringers are included in this object. Example: stairs, escalator, spiral staircases NOT interrupted with landings
Ramp	Inclined way or floor joining two surfaces at different levels. Example: Slope/Ramp interrupted with landings

Ramp Flight	Inclined slab segment, normally providing a human circulation link between two landings, floors or slabs at different elevations. Example: Slope/Ramp NOT interrupted with landings
Railing	The railing is a frame assembly adjacent to human circulation spaces and at some space boundaries where it is used instead of walls or to complement walls. It is designed to aid humans, either as an optional physical support, or to prevent injury by falling. Example: handrail, balustrade, grab bar, guard rail
Curtain Wall	Exterior wall of a building which is an assembly of components, hung from the edge of the floor/roof structure rather than bearing on a floor.
Roof	Construction enclosing the building from above Example: flat roof, gable roof, gambrel roof, hip roof
Building Element	The building element comprises all elements that are primarily part of the construction of a building. Example: structural and space separating system like walls, beams, doors
Site	A defined area of land, possibly covered with water, on which the project construction is to be completed. Example: terrain loose elements? exterior bench, vehicles, traffic elements?
BIM XReference	The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.
	Building Architecture Elements
Window	Construction for closing a (near) vertical opening in a wall or pitched roof that will admit light and may admit fresh air. The window is a building element that is predominately used to provide natural light and fresh air. A window consists of a lining and one or several panels. Example: skylights, light domes, swinging, pivoting, sliding, or revolving panels and fixed panels
Door	Construction for closing an opening. The door is a building element that is predominately used to provide controlled access for people and goods. A door consists of a lining and one or several panels. Example: hinged, pivoted, sliding, revolving, folding doors
Covering	A covering is an element which covers some part of another element and is fully dependent on that other element. Example: wall claddings, floorings, suspended ceilings, finish trim, base molding
Furnishing Element	Generalization of all furniture related objects. Example: closet, table, chair, bed
	Building Service Elements
Distribution Control Element	The occurrence elements of a building automation control system that are used to impart control over elements of a distribution system. It senses elements and measures changes in the controlled variable such as temperature, humidity, pressure, or flow. Example: thermostat
Distribution Port	The occurrence of a specialized port for use within the context of distribution elements. Example: A gas-powered hot water heater may have three ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member

	of two systems (GAS and DOMESTICCOLDWATER), and hosts one system (DOMESTICHOTWATER) at the corresponding port.

Distribution Flow Element	The occurrence elements of a distribution system that facilitate the distribution of energy or matter, such as air, water or power. Example: ducts, pipes, wires, fittings
Distribution Chamber Element	A place at which distribution systems and their constituent elements may be inspected or through which they may travel. Example: sump, trench
Energy Conversion Device	The occurrence of a device used to perform energy conversion or heat transfer and typically participates in a flow distribution system. Example: boiler, chiller, or a cooling coil
Flow Fitting	The occurrence of a junction or transition in a flow distribution system. Example: elbow, tee, a junction box in an electrical distribution system
Flow Controller	The occurrence of elements of a distribution system that are used to regulate flow through a distribution system. Example: damper, valve, switch, relay
Flow Moving Device	The occurrence of an apparatus used to distribute, circulate or perform conveyance of fluids, including liquids and gases, and typically participates in a flow distribution system . Example: pump, fan
Flow Segment	The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment. Example: air outlet, drain, water closet, sink
Flow Storage Device	Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow. Example: tank, battery
Flow Terminal	The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment. Example: air outlet, drain, water closet, sink
Flow Treatment Device	Device used to change the physical properties of the medium, flow treatment types (or the instantiable subtypes) may be exchanged without being already assigned to occurrences. Example: air, oil or water filter, duct silencer
	Building Structure Elements
Member	Structural member designed to carry loads between or beyond points of support. It is not required to be load bearing. The orientation of the member (being horizontal, vertical or sloped) is not relevant to its definition. Example: diagonal element of a truss construction
Footing	A part of the foundation of a structure that spreads and transmits the load directly to the soil. Example: wall footing, stepped footing, sloped footing, raft footing
Pile	A slender timber, concrete, or steel structural element, driven, jetted, or otherwise embedded on end in the ground for the purpose of supporting a load. A pile is also characterized as deep foundation, where the loads are transfered to deeper subsurface layers.

	Example: pile foundation
Reinforcing Bar	A steel bar, usually with manufactured deformations in the surface, used in concrete and masonry construction to provide additional strength. Example: reinforced concrete
Reinforcing Mesh	A series of longitudinal and transverse wires or bars of various gauges, arranged at right angles to each other and welded at all points of intersection. Example: reinforced concrete floor slab
Tendon	A steel element used to impart prestress to concrete when the element is tensioned. Example: wire, cable, bar, rod, strand
Tendon Anchor	A tendon anchor is the end connection for tendons in prestressed or posttensioned concrete.
Plate	A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.
Discrete Accessory	A discrete accessory is a representation of different kinds of accessories included in or added to elements. Example: visible steel corbel (corbel system made from steel components protruding from the master element), corner fixing plate (fixing plate attached to the corner of an element), steel plate (steel plate used as an accessory in a joint)
Fastener	Representations of fixing parts which are used as fasteners to connect or join elements with other elements. Example: jointing mortar, glue, weld
Mechanical Fastener	Fasteners connecting building elements mechanically. Example: bolt, screw, nail, nut, washer, rivet

Related Commands

BimAttachComposition - attaches a BIM composition to solids.
BimProperties - allows to create, edit and delete property definitions in a dialog box, and to organize properties in property sets.
BimSpatialLocations - opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.
Bimify - analyzes the model and runs automatic classification and spatial location assignment for the entire model.
Export - exports drawings in a variety of vector and raster formats, including *.ifc

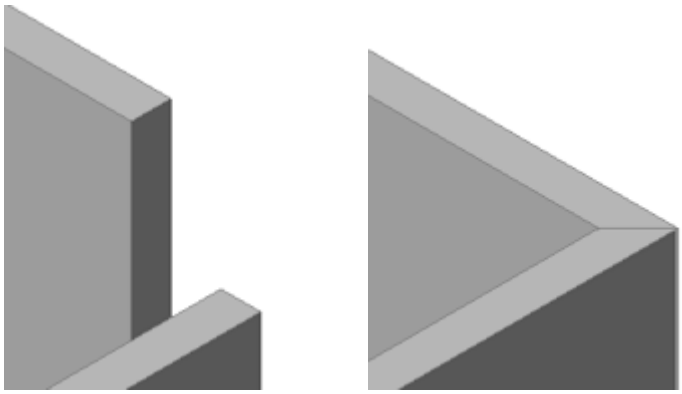
BimConnect

Creates an L-connection between minor faces of two solids.

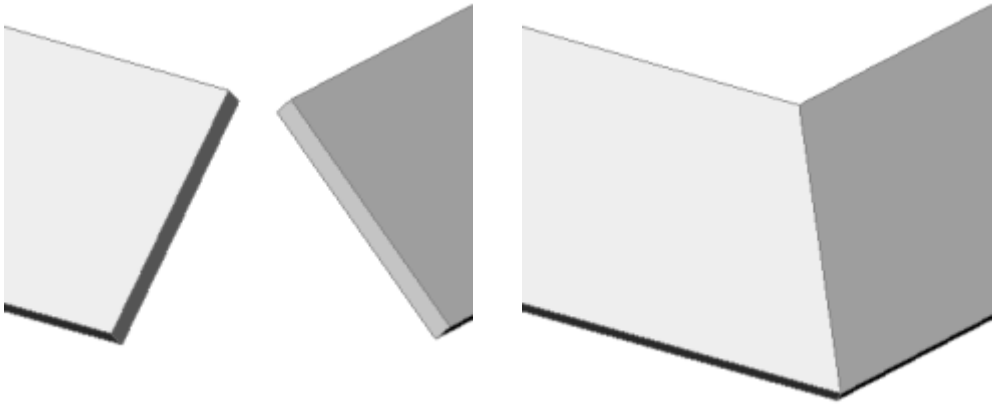
Accessing the Command

command bar: *bimconnect*
menu bar: *BIM | Connect*
toolbar: *BIM |*
Quad: *BIM |*

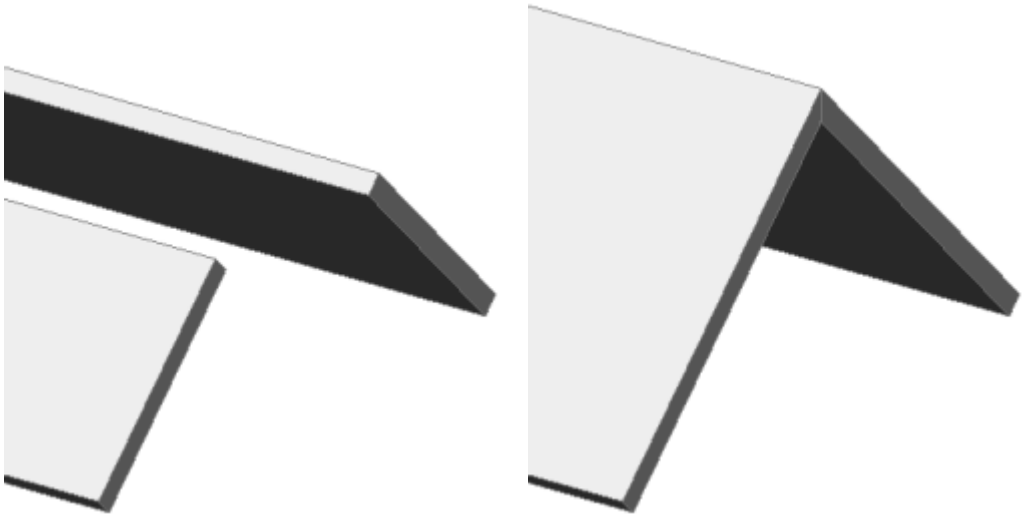
: ***bimconnect***
Prompts you in the command bar:
Select walls for connecting: (Select two walls.)
Accept connection <Ok>/Next: (Press Enter to create a bisector L-connection or choose an option.)



L-Connecting walls

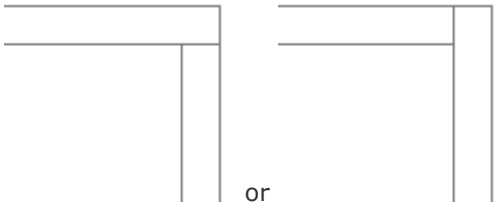
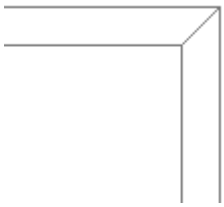


L-Connecting roof slabs



Connecting the ridge of roof slabs

Command Options

Option	Description
Ctrl-key	Hit the Ctrl key to see the next L-connection type. If TIPS is ON the currently selected connection type is indicated in the Tips widget. Press Enter to accept the current connection.
Next	Allows to choose between a parallel L-connection:  or a bisector L-connection:  Press Enter to accept the current connection.

Procedures

Creating L-Connections

Related Commands

BimDrag - drags major faces of solids while preserving connections with minor faces. Drags minor faces; optionally connects minor faces to major faces of other solids.

BimDrag

Drags major faces of solids: moves the solid while preserving connections with minor faces, optionally modifies the thickness or creates a copy of the solid.
Drags minor faces, optionally connects minor faces to major faces of other solids.
Allows to reposition a BIM section.

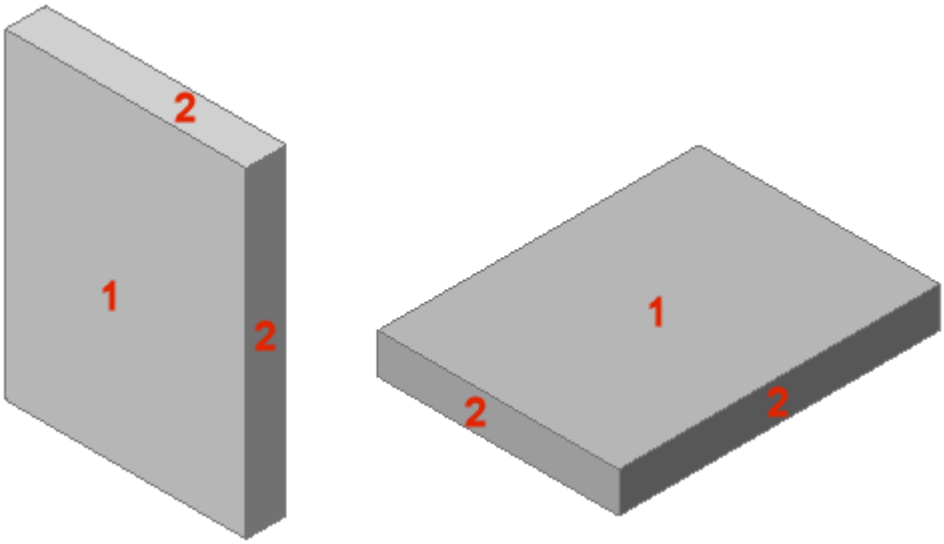
Accessing the Command

command bar: *bimdrag*
menu bar: *BIM | Drag*
toolbar: *BIM |*
Quad: *BIM |* or

: ***bimdrag***
Prompts you in the command bar:

Select several entities/subentities: *(select a face of a solid)*
Entities in set: 1

Select several entities/subentities: (select more faces, then right click or press Enter to stop selecting.)



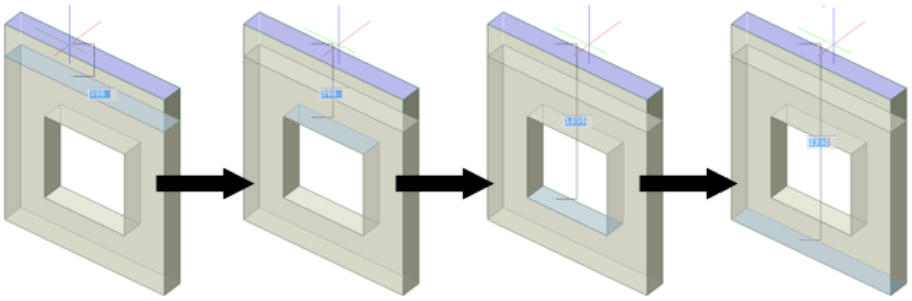
Major faces (1) and minor faces (2).

Command Options

Option	Description
Major faces	The options are: • Drag Solid: Moves the selected solid(s), while preserving connections with other solids. • Drag Face: Moves the selected face(s) only, thus changing the thickness (width) of the solid(s). • Copy Solid: Creates a copy of the selected solid(s). Prompts you:

	Specify distance to drag solid or [Switch to drag face]: Hit the Ctrl-key or type *S* to toggle between the options. If TIPS is ON the current mode is indicated in the *Tips* widget: • Drag Solid: • Drag Face: • Copy Solid: Do one of the following: • Specify a point in the drawing. • Type a value in the dynamic dimension field. Press the TAB key to select a different reference face. The current reference face highlights.
Minor faces	Moves the selected faces, thus stretches the solid; optionally connects the selected faces to the major face of another solid. Choose *Connect to Nearest* ( ) in the *Model* tab of the Quad to automatically connect the selected minor face to the nearest major face of another solid. Applying the *BimDrag* command to a minor face allows to: • Change the length of a wall. • T-connect a wall to another wall. • Change the height of a wall. • Connect a wall to a floor or roof slab • Change the dimensions of a slab. • Connect a slab to a wall. • Modify the dimensions of an opening. • Move a window or door vertically or horizontally, if applied to an inner face of the opening made by the insert. If TIPS is ON the *Tips* widget displays: • Move: • Connect: To move the selected face, do one of the following: • Specify a point in the drawing. • Type a value in the dynamic dimension field to move the face over a specfied distance. • Repeatedly hit the TAB key to select a different reference face to position the face with respect to a parallel face. The current reference face highlights. All faces which are parallel to the face being dragged are recognized, the nearest first. Press and hold the Shift key

when hitting the TAB key to cycle reference faces in reverse order.



Reference faces from left to right: top face of the solid / top face of the opening / bottom face of the opening / bottom face of the solid.

To connect the selected face,

- Press the Ctrl-key to connect to the major face of another solid.
  Prompts you:
  Select one or more 3d solid faces to connect <ENTER to auto-select>:

  Do one the following:

  - Press Enter to connect to the nearest face(s) of another solid or other solids.
  - Place the cursor over the face of the solid to connect to, the face highlights and the minor face being dragged connects to the face under the cursor. Click to select the face. Optionally, select more faces. Right click or press Enter to confirm.

Procedures

Creating T-connections
Creating T-connections ply-by-ply
Modifying the thickness of a solid
Modifying the height of a wall
Modifying the height of a wall ply-by-ply
Setting the distance between parallel walls
Creating a parallel copy of a wall
Creating a partial copy of a wall
Moving an insert vertically
Moving an insert horizontally

Related Commands

dmPushPull - adds or removes volume from a solid by moving a face.
dmMove - moves the selected solids, or faces or edges of a solid using a vector.

BimExport

Obsolete.
Use Export instead and select file type *.ifc.

BimFlip

Flips the starting face from which the layers of a composition are set out.
Mirrors an insert (e.g. a window or door) left/right or flips it inside/outside.

Accessing the Command

command bar: *bimflip*
menu bar: *BIM | Flip*
toolbar: *BIM* |
Quad: *BIM Classification* | (solids)
Quad: *BIM* | (inserts)

: ***bimflip***
Prompts you in the command bar:

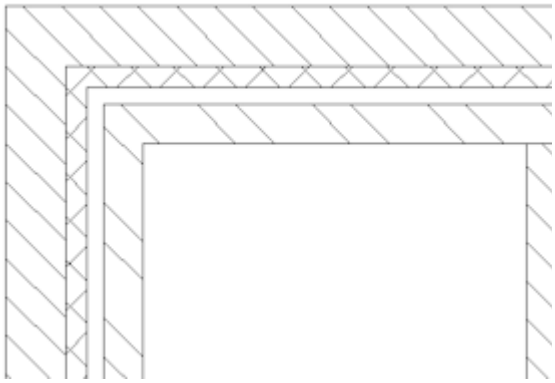
Select BIM entity to flip: (Select a wall, slab or an insert)
Entities in set: 1

Select BIM entity to flip: (Select more walls, slabs or inserts, or press Enter.)

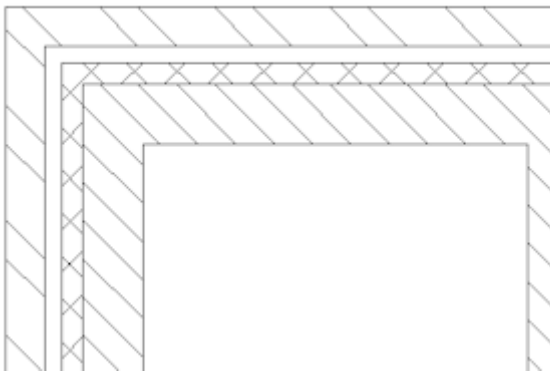
If the selection contains no inserts:

Done on xx object(s).

The result is visible in drawings generated by the BimSectionUpdate command if a multi-layered composition is attached to the sectioned solid.



Before



After

If the selection contains at least 1 insert:

Flip insert(s) Left-right/In-out <Left-right>: *(Press Enter to flip left-right or type I and press Enter to flip inside-outside.)*

Command Options

Option	Description
Select BIM entity to flip	The *BimFlip* command accepts inserts, walls or slabs. Solids that have no BIM classification yet, cannot be selected. The command only succeeds for solids that meet both of the following conditions: • have a BIM composition attached. • are in a calculated BIM section.

	BricsCAD reports the number of solids that could not be flipped: '... failed on xx object(s)'
Left-right	Mirrors the insert about a vertical axis in the face of the solid.
In-out	Flips the insert to the opposite face of the solid.

Related Commands

BimAttachComposition - attaches a BIM composition to solids.
BimClassify - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).
BimUpdateThickness - re-applies the overall thickness of a composition to the solid.
BimSection - creates a BIM section entity.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.

Bimify

Analyzes the model and runs automatic classification and spatial location assignment for the entire model. Creates new buildings and/or stories if necessary.
Section entities are converted to BimSection entities.

Accessing the Command

command bar: *bimify*
Quad: *BIM* | *Bimify*

: ***bimify***

Displays no prompts or dialog boxes.

Reports in the Prompt History panel:

BIM data assigned to 84 of 84 object(s), skipped 0 pre-assigned object(s)
CB6 classified as Wall
CB7 classified as Wall
CB8 classified as Wall
...
CC1 classified as Window
CC8 classified as Window
CC9 classified as Door
CCA classified as Window
...
CEC classified as Slab
CED classified as Slab
...
Building 'Building': <Created>
The spatial location has been assigned to 84 element(s).

Command Options

No command options.

Related Commands

BimAttachSpatialLocation - allows to attach or remove a spatial location to a selection of entities.
BimClassify - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).
BimSpatialLocations - opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.

BimInsert

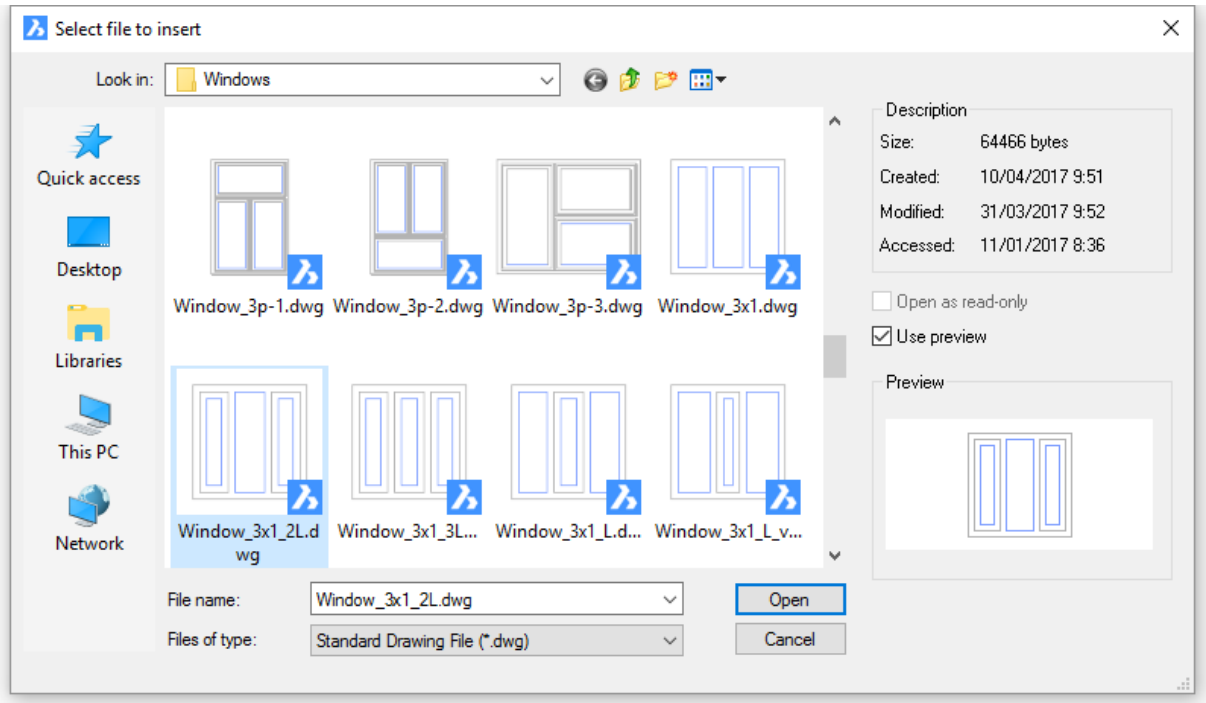
Inserts a window or door in a solid. Inserts a copy of a window or door in a solid.

Make sure Dynamic UCS (DUCS) and Dynamic Dimensions (DYN) are active.

Accessing the Command

command bar: *biminsert*
menu bar: *BIM| BIM Insert*
toolbar: *BIM |*
Qaud: *BIM |*

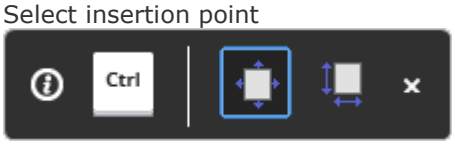
: ***biminisert***
Displays a File dialog box:

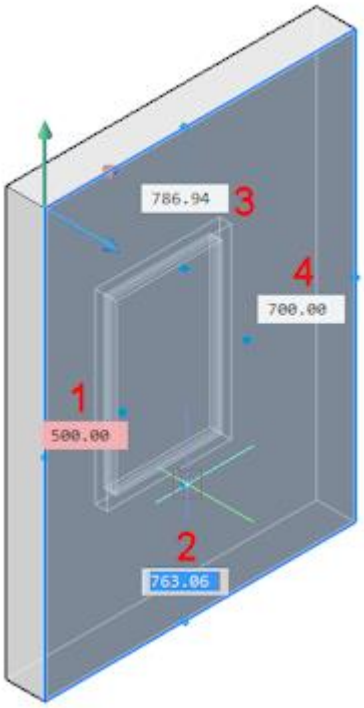
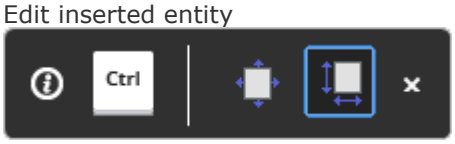


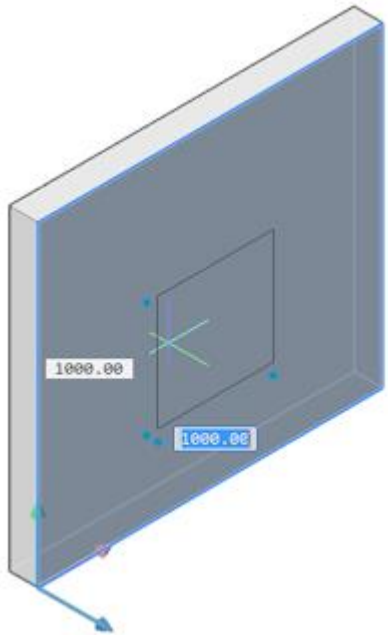
Select a window or door to insert, then double click or click the Open button.

Select insertion point or [Edit inserted entity] : (Move the cursor over a solid; the face under the cursor highlights: The UCS aligns dynamically with the selected face; if the edge by which the cursor enters the solid is horizontal or vertical, the X-axis of the UCS is horizontal, otherwise the X-axis is aligned with the edge; the Z-axis is perpendicular to the face of the solid. Specify point to insert.)
Hit the Ctrl-key to switch between Select insertion point and Edit inserted entity mode.

Command Options

Option	Description
Select insertion point 	Allows to specify the insertion point. Hit the Ctrl-key to switch to Edit inserted entity mode. From the middle of each edge of the placement rectangle dynamic dimensions display to the nearest parallel edge on the face of the solid.

	Press the TAB key to cycle the fields. Type a value in the dynamic dimension fields, then press the TAB key, to lock the field which then turns red. Only one of two parallel dimensions can be locked. If a dimension value is entered, and the other, parallel dimension was already locked, it will be unlocked.. Press Enter or click to insert the selected block.  1. Locked field 2. Current field 3 and 4. Unlocked fields
Edit inserted entity  	Allows to specify the width and height of the placement rectangle of the insert. Prompts you: **Edit [Height/Width/Done] <Height>:** Two dynamic dimension fields show the current width and height of the placement rectangle.

	 Type a value in the current dynamic dimension field, then press the TAB key to switch to the other field. Hit the Ctrl-key to switch to Select Insertion Point mode.

Procedures

Inserting windows or doors
Inserting a copy of an insert

Related Commands

BmUnlink - breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.
BimWindowCreate - creates a fully parametric window based on a closed contour.
BimWindowUpdate - updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

BimLinearSolid

Creates a chain of linear solids.

Accessing the Command

command bar: *bimlinearsolid*
menu bar: *BIM | Linear Solid*
toolbar: *BIM |*
Quad: *Model | Linear Solid*
Drag a profile from the *BIM Libaries - Profiles* panel onto the graphic screen.

: ***bimlinearsolid***
Prompts you in the command bar:

Set start point or [Follow/Rectangular/Circular/Dialog/select in Model/adJust profile] <Last point>:
(Pick a point or choose an option.)

Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:
(Pick a point or choose an option.)

Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:
(Pick more points, or choose an option, or press Enter to stop.)

Command Options

Option	Description
Last point	Press Enter to use the most recently specified point.
Follow	Draws the next linear solid at the same angle as the previous one. Length of line - specify the length of the linear solid.
Rectangular	Switches to a default rectangular profile.
Circular	Switches to a default circular profile.
Dialog	Opens the Profiles dialog box. Select a profile, then click the OK button on the Profiles dialog box. Returns to the previous prompt.
Select in model	Allows to select a profile in the model; prompts you: Pick a profile - Do one of the following: • Click a closed 2D entity or a region in the model. • Click an existing linear solid. Returns to the previous prompt.
Adjust profile	Allows to adjust the dimensions of the default rectangular and circular profile; prompts you: For rectangular profile: Width of rectangle <250> [Height] - Type a value or press Enter to accept the current value. Height of rectangle <120> [Width] - Type a value or press Enter to accept the current value. For circular profile: Radius of circle <100> - Type a value or press Enter to accept the current value. Returns to the previous prompt.

Angle	Draws at a user-specified angle and distance; prompts you: Angle of line - specify the angle (measured counter-clockwise from the positive x-axis). Length of line - specify the length of the linear solid.
Length	Draws at a user-specified distance and angle; prompts you: Length of line - specify the length of the linear solid. Angle of line - specify the angle (measured counterclockwise from the positive x-axis).
Undo	Erases the last linear solid.
Quarter turn	Rotates the profile 90° counter clockwise.
Rotate	Allows to rotate the profile by a user defined angle: prompts you: Angle to rotate profile - specify the rotation angle. Positive values rotate the profile counter clockwise.
Choose other profile	Allows to use a different profile for the next linear solid; prompts you: Choose a profile [**Rectangular**/**Circular**/**Dialog**/select in Model] - choose an option.
Adjust profile	Prompts you to pick a profile for the next linear solid in the model: Pick a profile - Do one of the following: • Click a closed 2D entity or a region in the model. • Click an existing linear solid.

Related Commands

BimAddEccentricity - allows to control the relative position of the axis of a linear solid.
BimApplyProfile - applies a profile to linear entities and linear solids.
BimProfiles - opens the profiles library in the current project and the referenced library.

BimMultiSelect

Selects linear solids with coplanar and/or parallel axes.

Accessing the Command

command bar: *bimmultiselect*
menu bar: *BIM | multiselect*

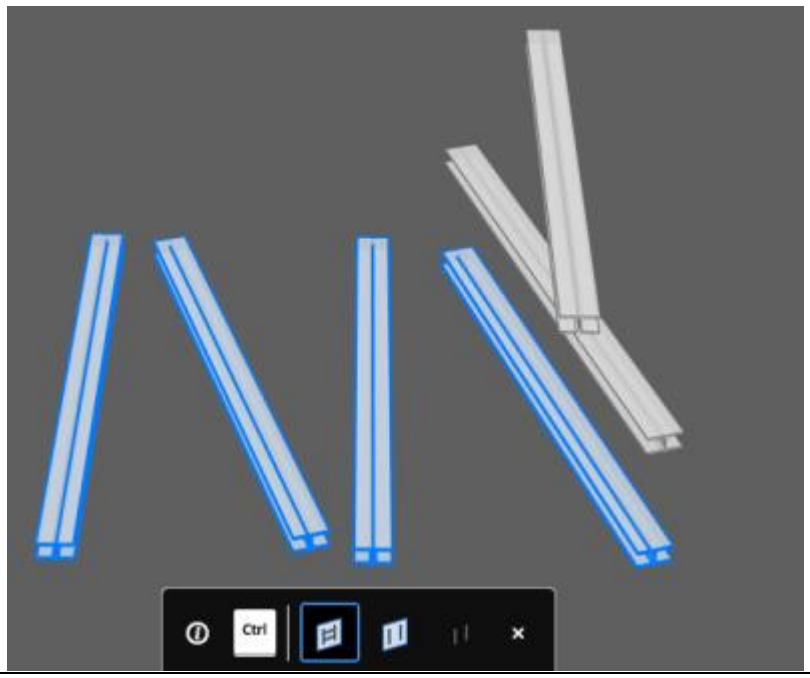
: ***bimmultiselect***
Prompts you in the command bar:

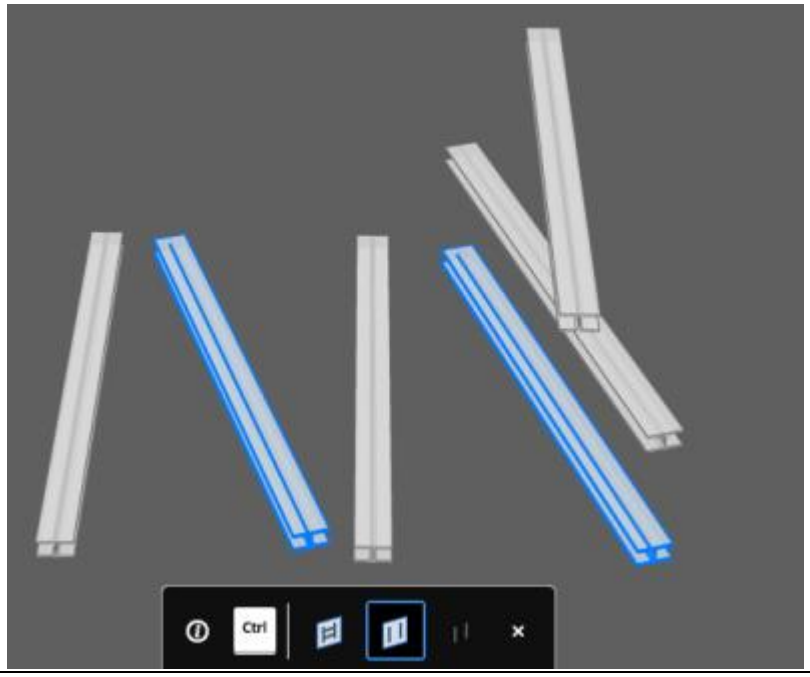
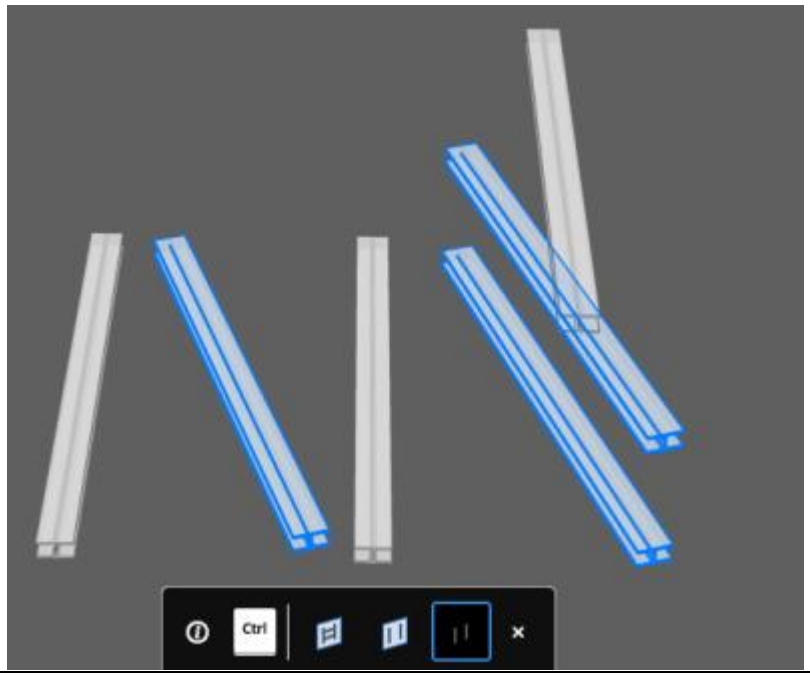
Select single entity/subentity: (Select a solid or a solid face.)

Select type of selection [all Coplanar axes/coplAnar and parallel axes/all Parallel axes] <Accept>: (Choose an option or press Enter to accept selection of the highlighted linear solids.)

If a face of a linear solid is selected, the corresponding faces of the linear solids with coplanar/parallel axes are selected.

Command Options

Option	Description
All coplanar axes	Selects all solids of which the axis is coplanar with the axis of the selected solid. 

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. 

Related Commands

dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelectAlignedFaces - selects all faces in a model which are coplanar with a selected face.
SelectAlignedSolids - selects all solids in a model of which a face is coplanar with a selected face.
SelectConnectedFaces - selects all faces in a model which are connected to a selected face.

SelectConnectedSolids - selects all solids in a model which are connected to a selected face.

BimPatch

Allows to replace part of a generated section.
The command is valid in model space only.

Accessing the Command

command bar: *bimpatch*

: ***bimpatch***
Prompts you in the command bar:
Specify start point of rectangle (in Model Space): (Specify a point.)

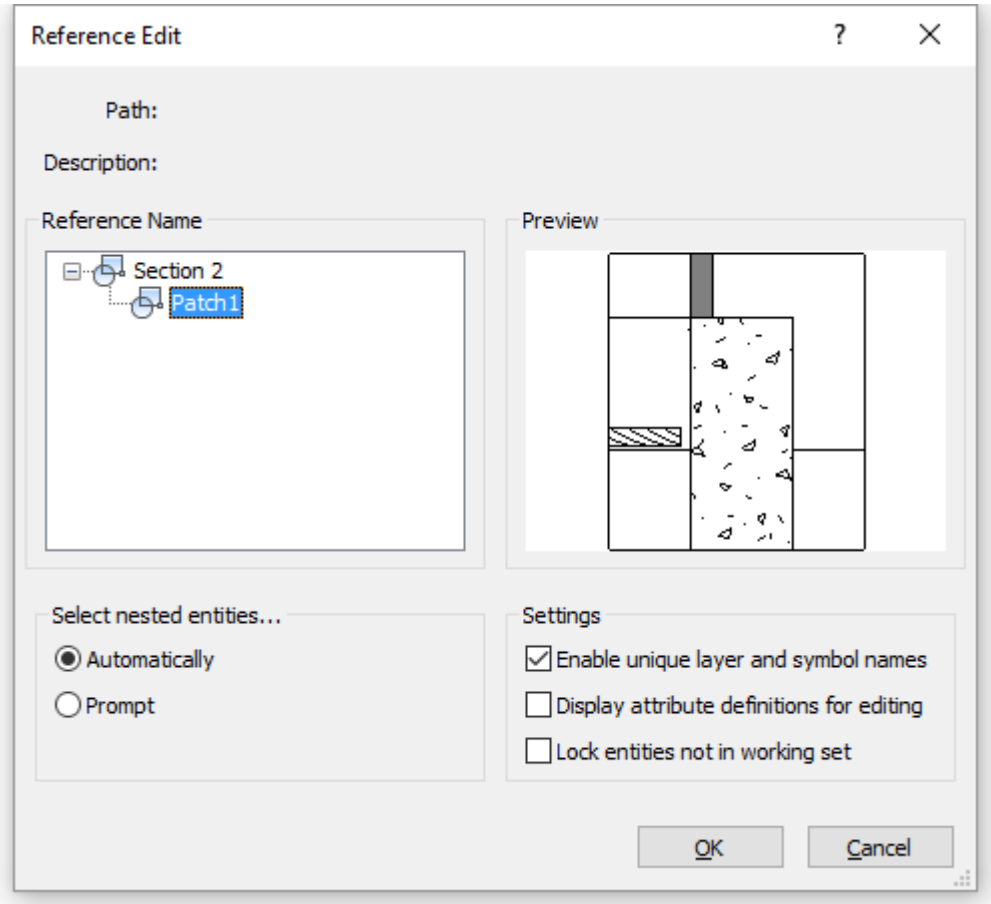
Specify end point of rectangle: (Specify a point or use the dynamic dimension fields to specify the dimensions of the rectangle).

NOTE If BricsCAD responds: 'Cannot apply patch: no intersection points found' set the GENERATEASSOCVIEWS system variable ON.
Lookup the variable in the Settings dialog under **Dimensions** or type GENERATEASSOCVIEWS in the command bar and choose ON.

A cut-out is generated as a nested block in the BIM section. BIM patch blocks are named Patch1, Patch2, ...

The rectangular boundary of a BIM patch is placed on layer BIMPATCH_BOUNDARY.

Double click the BIM section to launch RefEdit.



Select the BIM Patch in the *Reference Name* tree, then click the *OK* button.

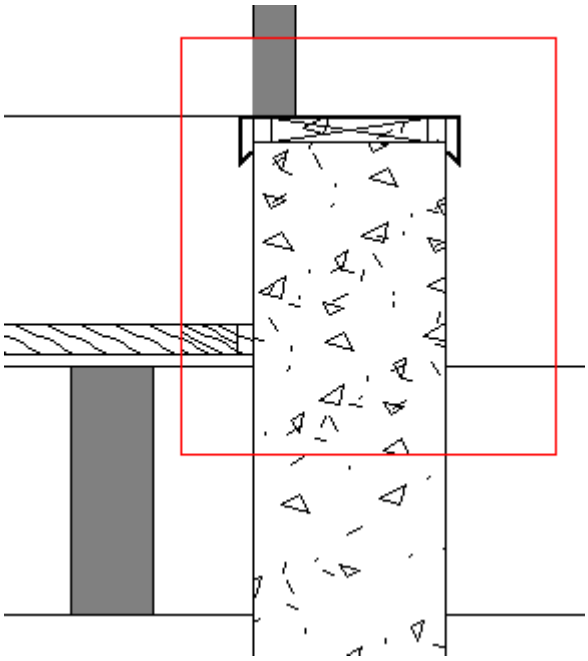
The *Reference Edit* dialog box closes.

Edit the BIM Patch.

Use RefClose to save the changes in the BIM Patch.

When the corresponding BIM section is updated, BricsCAD will check if the generated section result within that rectangle is identical to the result which was generated at the time the BIM patch was created:

- If not, then the user edited BIM patch is preserved but the boundary is colored red.
- If it is, the user edited BIM patch is preserved.



Intersection points and endpoints of curves originating from 3d objects are used to find the position of a BIM patchrectangle. When objects are modified or moved in the 3d model in such way that the position of these intersection points change, but the geometry inside the BIM patchrectangle is not affected, then BricsCAD will relocate the user edited BIM patch.

Command Options

No command options

Related Commands

BimSection - creates a BIM section entity.
BimSectionOpen - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.
RefEdit - dits block references and externally-referenced drawings through a dialog box.

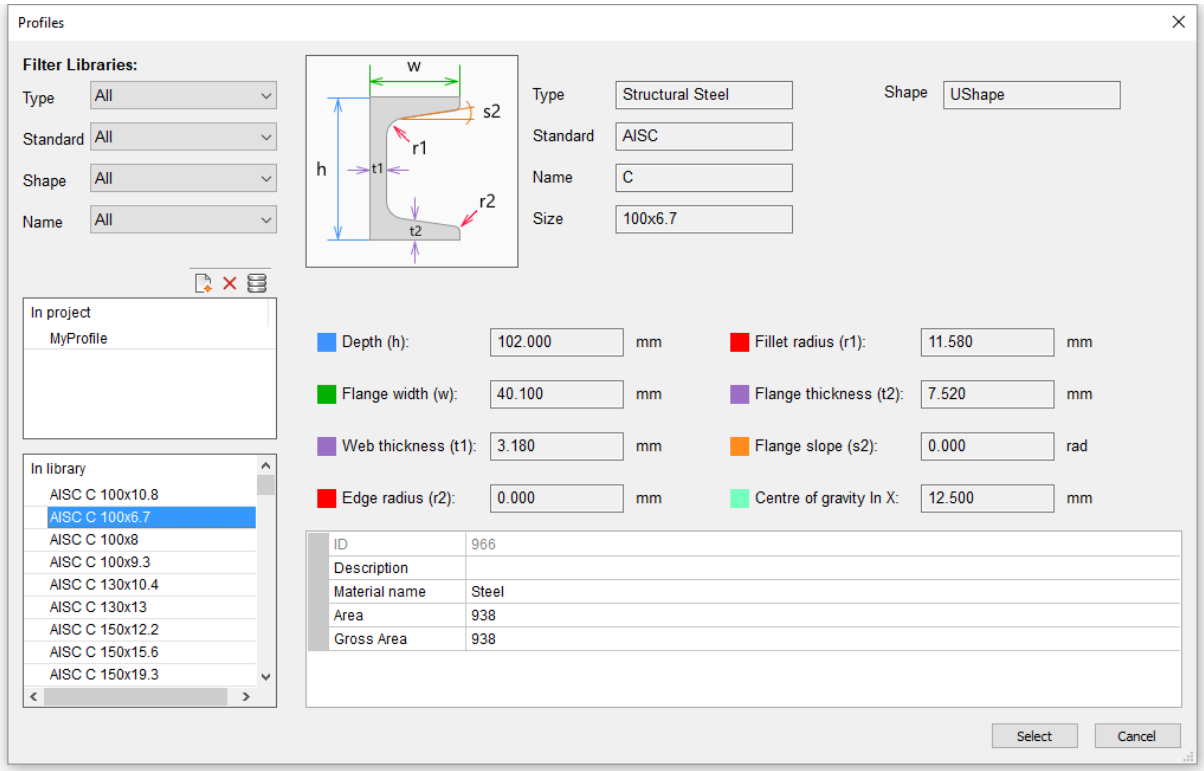
BimProfiles

Opens the profiles library in the current project and the referenced library.

Accessing the Command

command bar: *bimprofiles*
Bim Libraries panel: *Profiles Toolbar |*

: ***bimprofiles***
Displays a dialog box:

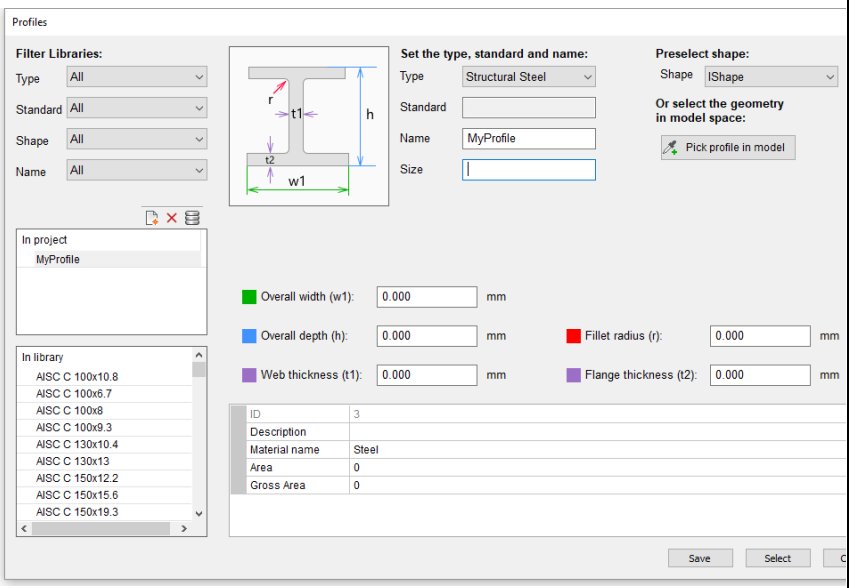


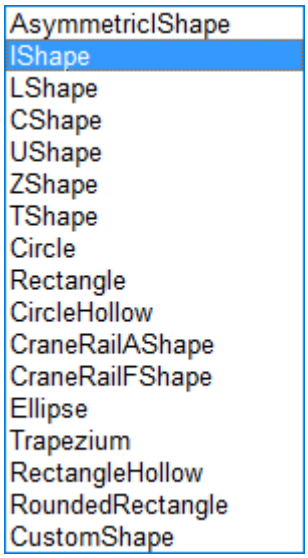
Select a profile in the In project or In library list to edit its properties.

Drag and drop profiles to copy them from the Library to the Project and vice versa.

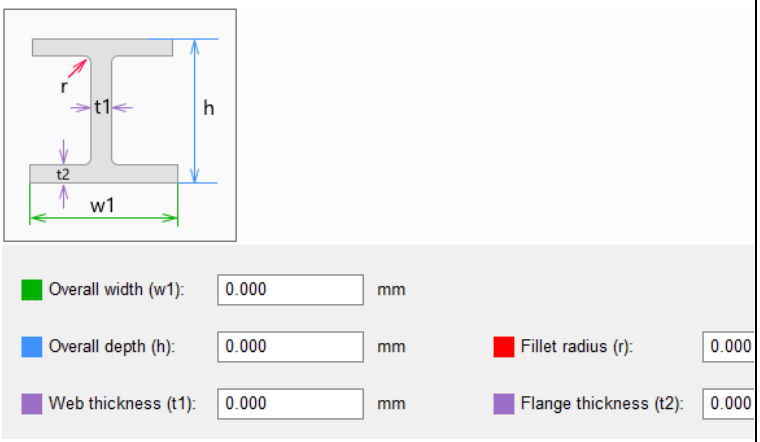
Command Options

	Description
Filters	Allows to filter the profiles list by *Type*, *Standard*, *Shape* and *Name*.
Type All	The *Type* options are: *All*, *Generic*, *Structural Concrete*, *Structural Steel*.
Standard All	The *Standard* options are: *All*, *AISC* (Americal Institute of Steel Consctruction), *BS* (British Steel), *EURO* (European standard steel profiles), *User Defined.*
Shape All	The *Shape** options are: *All*, *Circle Hollow*, *I Shape*, *L Shape*, *Rectangle*, *Rectangle Hollow*, *T Shape*, *U Shape* *Depending on the selected *Standard* some *Shape* options are not available.

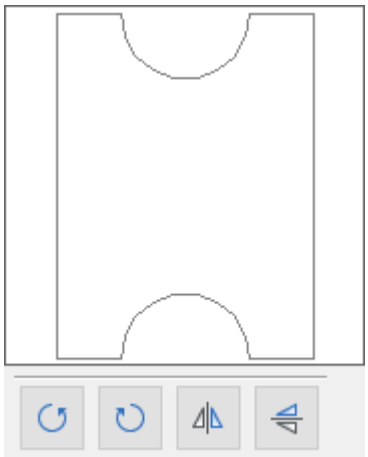
Name All	The *Name* options depend on the selected *Type*, *Standard* and *Shape* option. E.g. available options for *Structural Steel / AISC / I Shape* are: HP, M, S, W.
Tools	
	Creates a new profile.  • Type: Select a type. All Generic Structural Concrete Structural Steel HVAC Piping Electrical • Name: Type a name • Size: Indicate the size of the profile. • Shape: Do one of the following: • Select a shape.



Fill out the properties grid for the selected profile:



- Click the *Pick Profile in Model* button.
  Prompts you: Pick a profile - the *Profiles* dialog box temporarily closes to let you select a closed 2D entity or the boundary entities of an enclosed area.



Optionally, click the buttons at the bottom of the profile preview to rotate or mirror the profile.

✕	Deletes the selected profile
New Profile Duplicate Delete Copy to Project	Select a profile, then right click to display the context menu: • New Profile: Creates a new profile. • Duplicate: Creates a new profile as a duplicate of the selected profile. • Delete: Deletes the selected profile. • Copy to Project: Copies the selected profile to the Project library.

Related Commands

BimAddEccentricity - allows to control the relative position of the axis of a linear solid.
BimApplyProfile - applies a profile to linear entities and linear solids.
BimLinearSolid - creates linear solids.
BimRecalculateAxis - recalculates the axis of linear building elements and repositions it back on the center line of a linear element.

BimProjectInfo

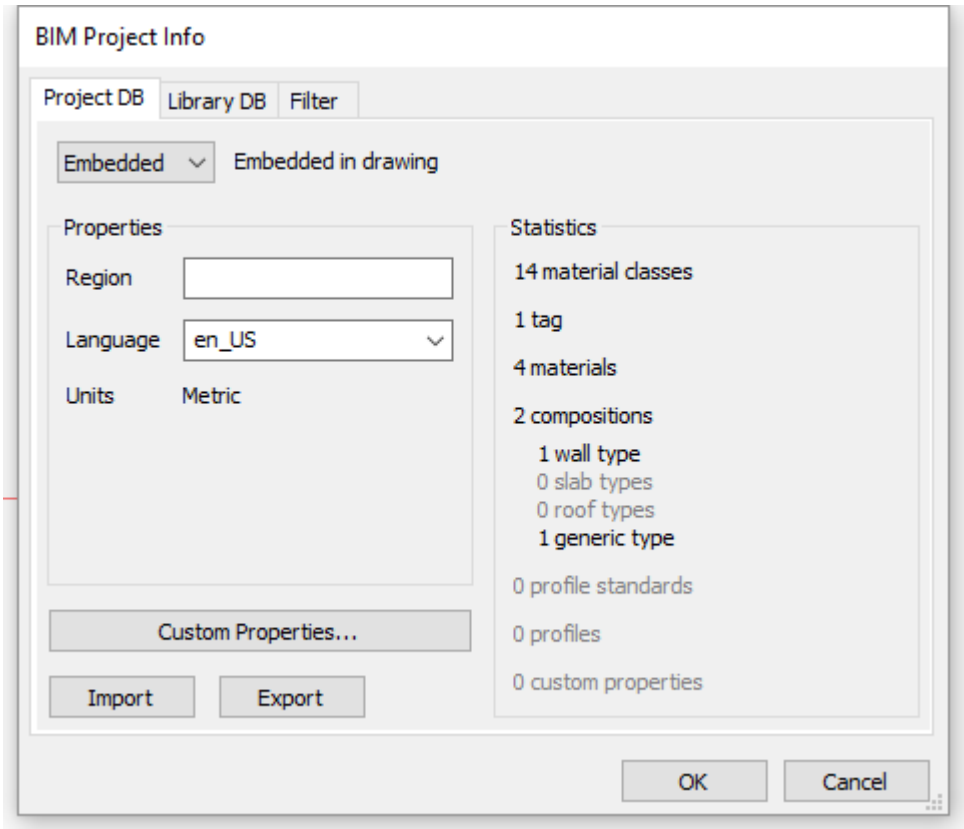
Opens the BIM Project Info dialog box.

Accessing the Command

command bar: *bimprojectinfo*

: ***bimprojectinfo***

Displays a dialog box:



Procedures

To set the library database
To set the project database
To set the project filter

Related Commands

BimProfiles - opens the profiles library in the current project and the referenced library.
BLCompositions - opens the compositions library in the current project and the referenced library.
BLMaterials - opens the physical materials library in the current project and the referenced library.

BimProperties

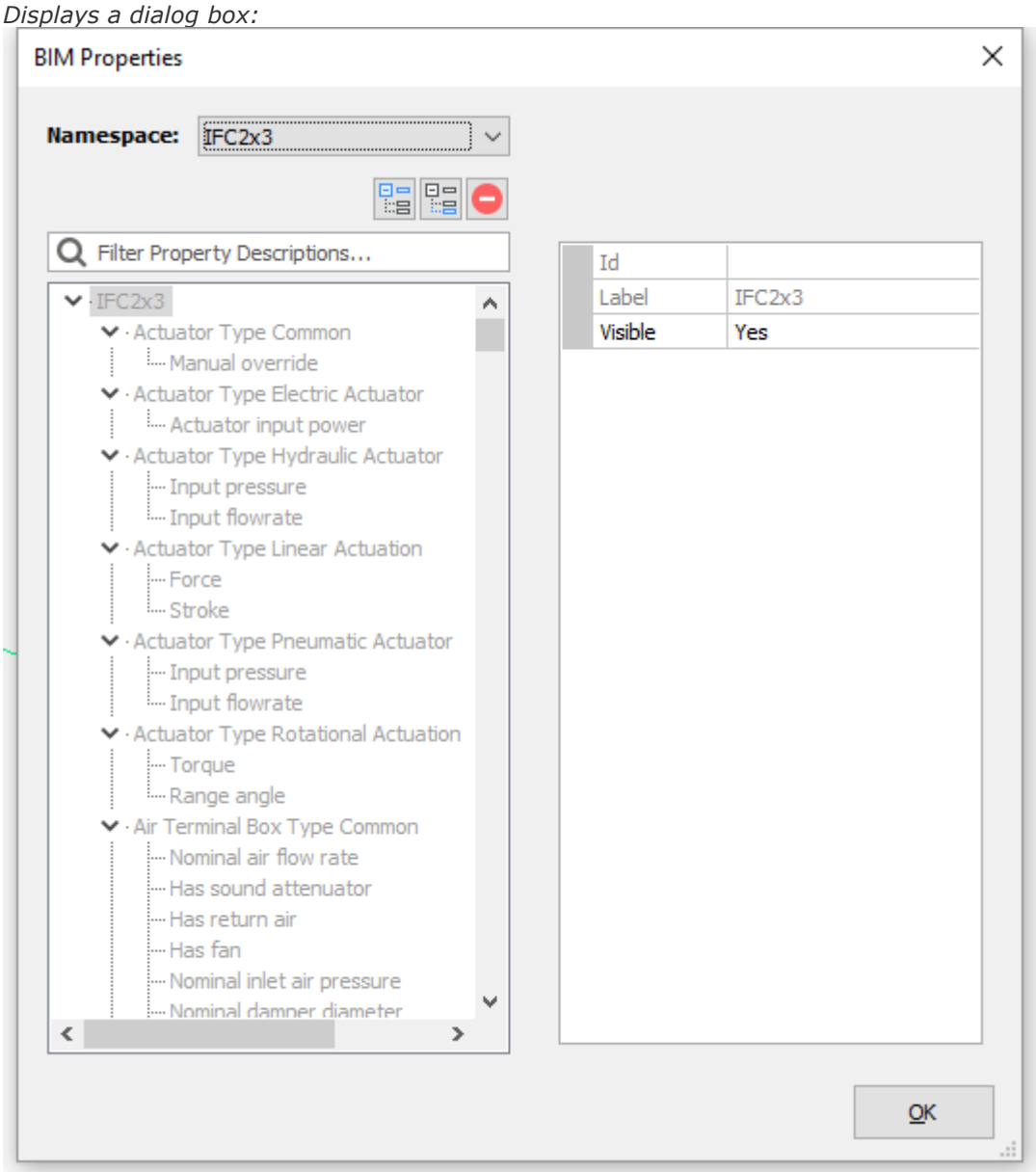
Allows to create, edit and delete property definitions in a dialog box, and to organize properties in property sets.

BIM properties are included in IFC export.

Accessing the Command

command bar: *bimproperties*

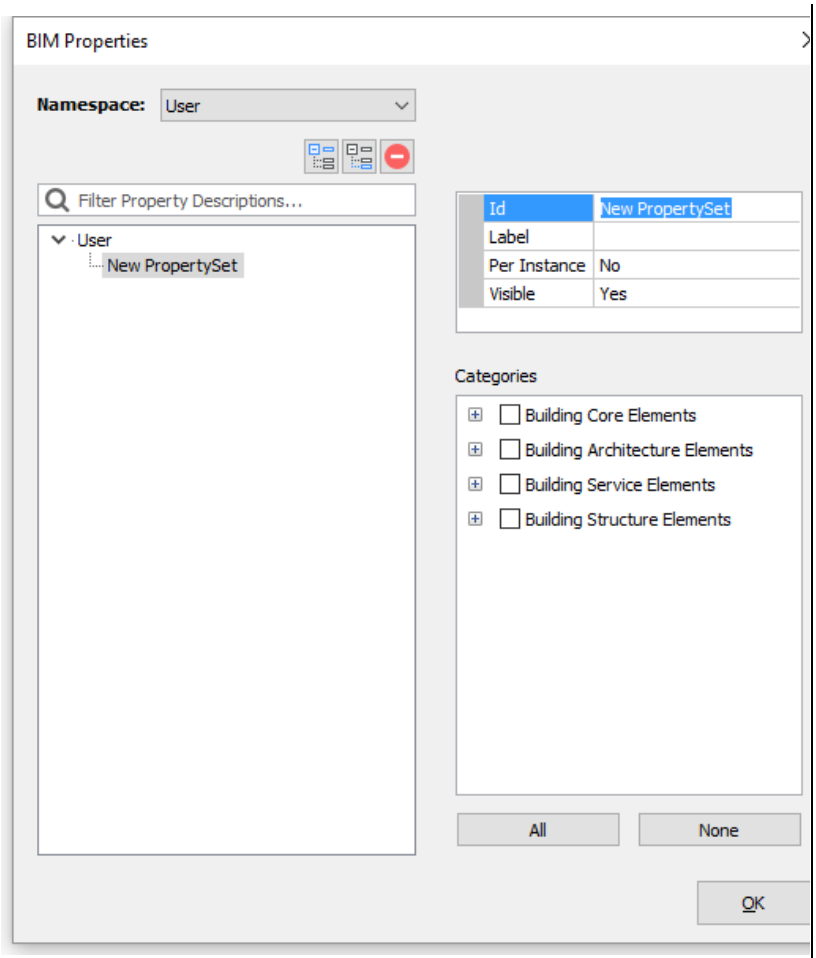
: ***bimproperties***
Displays a dialog box:



The BIM properties are saved in the **.bsyslib* project database. See Project Info in the User Guide.

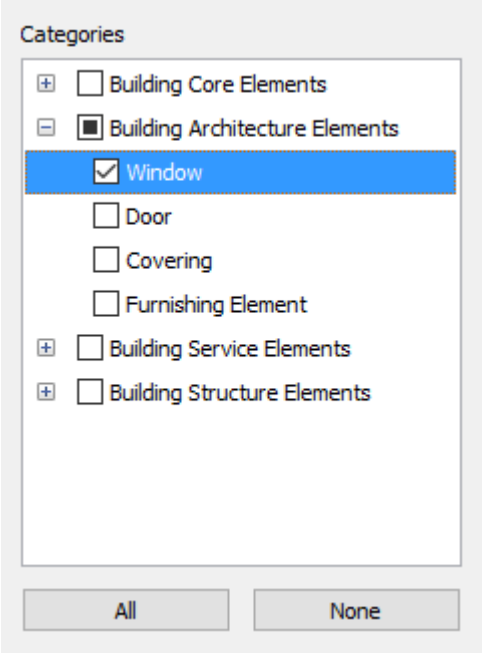
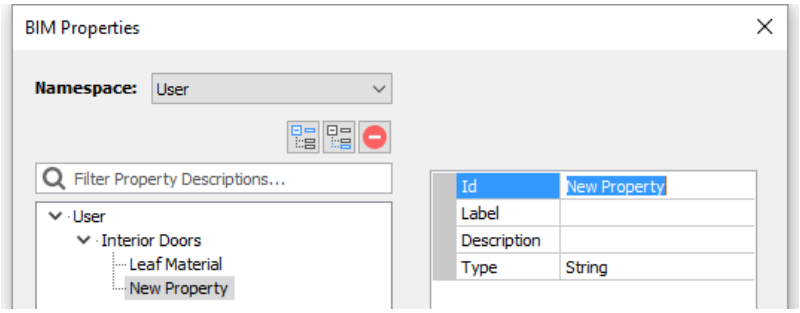
Command Options

Option	Description
Namespace	Properties exist in a namespace. Two namespaces are available: **IFC2x3**: IFC properties are read-only. Id Label: IFC2x3 Visible: Yes • **Id**: This field is empty and cannot be edited. • **Label**: This field cannot be edited. • **Visible**: Defines whether the IFC2x3 properties are visible in the Properties panel. **User**: Id: User Label: My Properties Visible: Yes • **Id**: Name of the userspace. This field is not editable. • **Label**: The label of the namespace is used in the properties tree. If the Label field is empty, the Id is used instead. • **Visible**: Defines whether the namespace properties are visible in the Properties panel.
Filter Property Descriptions...	Allows to filter the property tree. Type a search string in the field. Only properties of which the *Id* or the *Label* contains the search string display. The properties tree is filtered as you type.
IFC2x3 namespace	
	The IFC2x3 properties cannot be edited. You can only control the visibility of the various property sets. To control the visibility of an IFC property set: • Select IFC2x3 namespace. • Select the property set. • Set the *Visible* property of the property set to *Yes*.
User namespace	
Add Property Set	Allows to add a property set in the current namespace. Click the button to add a new property set.



Do the following:

- Type a name in the *Id* field.
- Type a name in the *Label* field.
  The *Label* name appears in the properties tree. If the *Label* field is empty, the *Id* name displays.
- Set the *Per Instance* property (not implemented yet).
- Set the *Visible* property.
  Defines whether the properties in the property set are visible in the Properties panel.
- Select the building elements the property set applies to. All entities which are classified this way get the property assigned automatically.
  Available categories and building elements are the same as in the BimClassify command.

	 • Expand a category, then check building elements. • Click a category checkbox to select all building elements in that category • Click the *All* button to select all building elements in all categories. • Click the *None* button to uncheck all building elements in all categories.
Add Property	Allows to add a property to the currently selected property set. Select a property set in the properties tree, then click the button to add a new property.  Do the following: • Type a name in the *Id* field. • Type a name in the *Label* field. The *Label* name appears in the properties tree. If the *Label* field is empty, the *Id* name displays. • (optional) Type a description in the *Description* field. • Select the property Type. The options are: *Boolean*, *Integer*, *Real* and *String*.
Delete	Deletes the selected Property or Property set.

Related Commands

BimExport - saves a BIM model as an IFC file for use with an IFC-certified applications.
BimClassify - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).

BimRecalculateAxis

Recalculates the axis of linear building elements and repositions it back on the center line of a linear element.

Accessing the Command

command bar: *bimrecalculateaxis*
Quad: *Model*

: ***bimrecalculateaxis***
Prompts you in the command bar:

Select several entities/subentities: **(Select a linear solid.)**
Entities in set: 1

Select several entities/subentities: **(Select more linear solids or press Enter.)**

Command Options

No command options.

Related Commands

BimAddEccentricity - allows to control the relative position of the axis of a linear solid.

BimReposition

Repositions an insert in the face of a solid.

Accessing the Command

command bar: *bimreposition*
menu bar: *BIM | Reposition*
toolbar: *BIM |*
Quad: *BIM |*

: ***bimreposition***
Prompts you in the command bar:

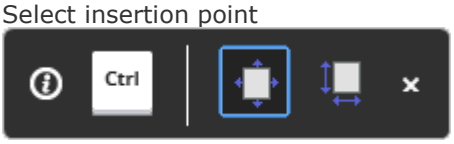
Select an insert to move: (select an insert)

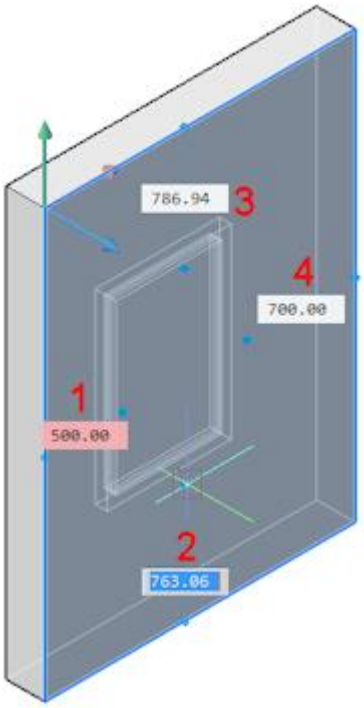
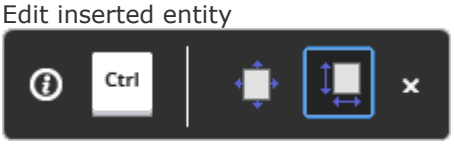
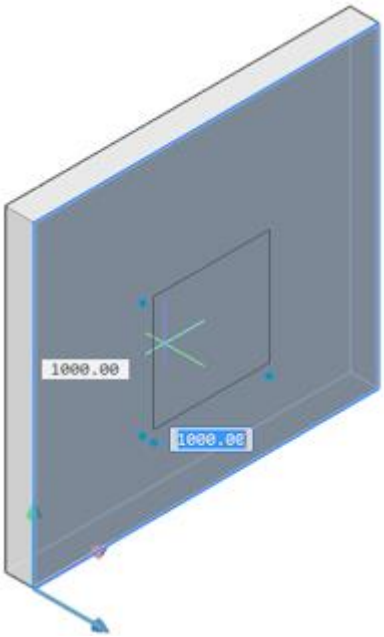
The selected insert moves dynamically in the face of the solid. From the middle of each edge of the placement rectangle dynamic dimensions display to the nearest parallel edge on the face of the solid.

Select insertion point or [Edit inserted entity]: (Specify an insertion point or type a value in the dynamic dimension fields; press the TAB key to cycle the dynamic dimension fields.)

Hit the Ctrl-key to switch between Select insertion point and Edit inserted entity mode.

Command Options

Option	Description
Select insertion point 	Allows to specify the insertion point. Hit the Ctrl-key to switch to Edit inserted entity mode. From the middle of each edge of the placement rectangle dynamic dimensions display to the nearest parallel edge on the face of the solid. Press the TAB key to cycle the fields. Type a value in the dynamic dimension fields, then press the TAB key, to lock the field which then turns red. Only one of two parallel dimensions can be locked. If a dimension value is entered, and the other, parallel dimension was already locked, it will be unlocked.. Press Enter or click to insert the selected block.

	 1. Locked field 2. Current field 3 and 4. Unlocked fields
Edit inserted entity 	Allows to specify the width and height of the placement rectangle of the insert. Prompts you: **Edit [Height/Width/Done] <Height>:** Two dynamic dimension fields show the current width and height of the placement rectangle.  Type a value in the current dynamic dimension field, then press the TAB key to switch to the other field.

	Hit the Ctrl-key to switch to Select Insertion Point mode.

Procedures

Repositioning an insert

Related Commands

BimInsert - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.
DmMove - moves the selected solids, or faces or edges of a solid, or inserts using a vector.
BimWindowUpdate - updates the opening made in a solid by an insert, in case the definition changed in such way that the opening was not correctly updated automatically.

BimRoom

Creates BIM Room entities.

Accessing the Command

command bar: *bimroom*
menu bar: *BIM | Room*
toolbar: *Bim |*

: ***bimroom***
Prompts you in the command bar:

Pick a point inside the room or <select>: (Pick a point inside a space enclosed by solids. The enclosing solids must be classified as walls. See BimClassify.)

A label is created at the geometric center of the room. Use the BimTag command to add the room label to a plan section.

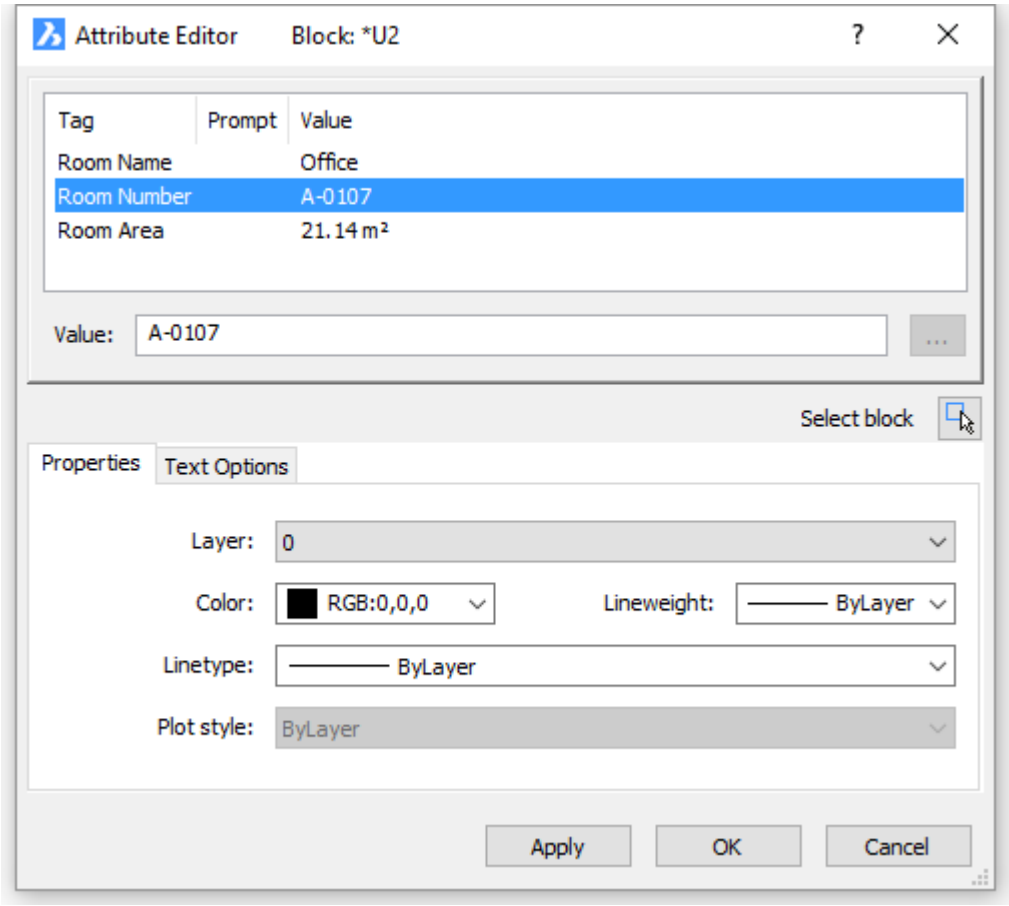
Command Options

Option	Description
Pick a point	Creates a BimRoom entity from an enclosed space*. Prompts you: Pick a point or [select 3d Solid]: Pick a point inside the room area. Dynamic UCS defines the bottom plane of the room. In this plane, an area enclosed by walls is found and a room mark is placed. The room mark is a block consisting of a hatch and attributes for name, number and area. The enclosing solids must be classified as walls. See BimClassify. A room stamp is created in the center of the room. You can control the layout of the room stamp by editing _RoomCallout.dwg in the *Support\Bim\Sections* subfolder of the Roamable Root folder. By default this is: *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support\Bim\Sections*. *Boundary entities must be Bim classified. To divide a large space in different rooms, divide the space using linear entities, which you then classify as Walls.
Select	Creates a static BimRoom entity from an existing entity or a detected boundary. Prompts you: Select 3d solid, region, polyline, or boundary - Select an entity or click inside an enclosed area*. * The *Enable Boundary Detection* option ( ) of the SELECTIONMODES system variable must be selected.
Room Properties	A Room corresponds to an IfcSpace, and has all IFC properties, apart from the BIM properties. Room properties can be edited in the Properties Bar:

Attributes	
Room Name	Office
Room Number	1.003
Room Area	11.19 m²
BIM	
Type	Room
Name	Office
Description	
Building	Sky Office & Housing
Story	Ground Floor
GUID	3bXiwHzobCcgLVkAk4zMCd
Number	1.003
Department	
Wall north finish	Facing Bricks, Hand-formed
Wall east finish	Gypsum Board
Wall west finish	Gypsum Board
Wall south finish	Gypsum Board
Room representation	Solid
Interior Elevations	Off

- Building: Click the field, then select a building (see BimSpatialLocations).
- Story: Click the field, then select a story (see BimSpatialLocations).
- Room representation: The options are
  - None: Room stamp only.
  - Hatch: Room stamp and a solid hatch.
  - Solid: 3D solid. The height of the solid is copied from the surrounding wall solids.
- Interior Elevations: Controls the display of *Interior Elevation* sections.
  If interior elevations are not created yet for the room the property is disabled.
  (see BimSection).

or using the EAttEdit command:



Related Commands

BimUpdateRoom - updates the geometry of a Room entity.
BimTag - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

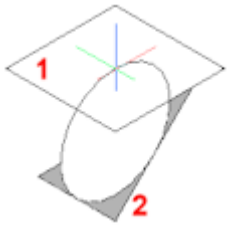
BimSection

Creates a BIM section entity.

Accessing the Command

command bar: *bimsection*
menu bar: *BIM | Define Section*
toolbar: *BIM |* *or* (Detail Section)

: ***bimsection***
The Section Tracker is attached to the cursor.



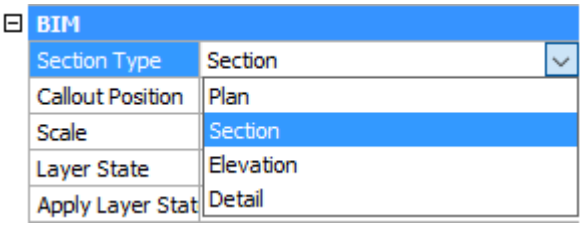
The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative Z-direction of the current coordinate system.
If Dynamic UCS (UCSDETECT) is ON, the section plane aligns to the face of a 3D solid under the cursor.

Prompts you in the command bar:
Select a point to place section or [Detail/Interior/Scale]: *(Specify a point.)*
The section plane displays dynamically parallel to UCS XY-plane, through the cursor position.
The clip display property of the newly created section plane is ON by default.
Specify distance: *(Type a distance* or specify a point.)*
The section plane is defined through the point or at the specified offset from the first point.

* It is recommended to have Dynamic Dimensions (DYN) active, which allows to type the distance in the dynamic entry field.

The section line is parallel to the X-axis of the UCS or dynamic UCS and through the point at the specified offset from the first point.
Only the section line and the callouts of a BIM Section entity display. When highlighted or selected the section plane, section boundary and/or section volume display.
The *Clip Display* property of the BIM Section entity is enabled by default. Double click the section line or a callout of a section to toggle *Clip Display* on/off. Multiple sections can have *Clip Display* on simultaneously.
The *Clip Display* property can be saved in a model view (See the View command).

BIM section entities are created on the current layer. It is recommended to create a dedicated layer for BIM sections. The *BIM_Sections* layer is created automatically along the first BIM section entity.
Horizontal sections get the *Plan* section type, other sections get the *Section* section type, unless no geometry is cut by the section plane, then the section type is *Elevation*.
The section type can be edited in the *Section Type* field on the *Properties Bar*.



NOTE The GENERATEASSOCVIEWS system variable controls whether VIEWSECTION produces 2D drawings which are persistently associated with the source 3D model.

If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified and BIMSECTIONUPDATE is executed.

Command Options

Option	Description
Scale	Sets the *Scale* property of the viewport in the drawing file created by the BIMSECTIONUPDATE command. The default *Scale* is saved through the *SectionScale* user preference. Prompts you: **Specify a scale for the viewport with the section result (0.000001 - 1000000.000000) <0.020000>:** Do one of the following: • Press Enter to accept the default scale. The default scale is saved through the SectionScale user preference. • Type a value, then press Enter. The value is saved to the SectionScale user preference.
Detail	Creates a *Detail* section type. Prompts you: Select first corner of the volume box base rectangle or [Based on an existing section/Scale]: Specify a point, or choose an option. Select the oppersite corner of volume box base rectangle: Do one of the following: • Specify a point. • Use the dynamic dimension fields to define the base rectangle. Hit the TAB key to switch between the dynamic dimension fields. **Select the height of volume box:** Do one of the following: • Specify a point. • Type a value in the dynamic dimension field.
Based on an existing section	Creates a *Detail* section type based on an existing section. The base rectangle of the detail section will be parallel to the section plane of the selected section. Prompts you: **Select an existing section plane as base:** Select a Bim section. Select first corner of the volume box base rectangle or [Based on an existing section/**Scale**]: Specify a point, or choose an option. Select the oppersite corner of volume box base rectangle: Do one of the following: • Specify a point.

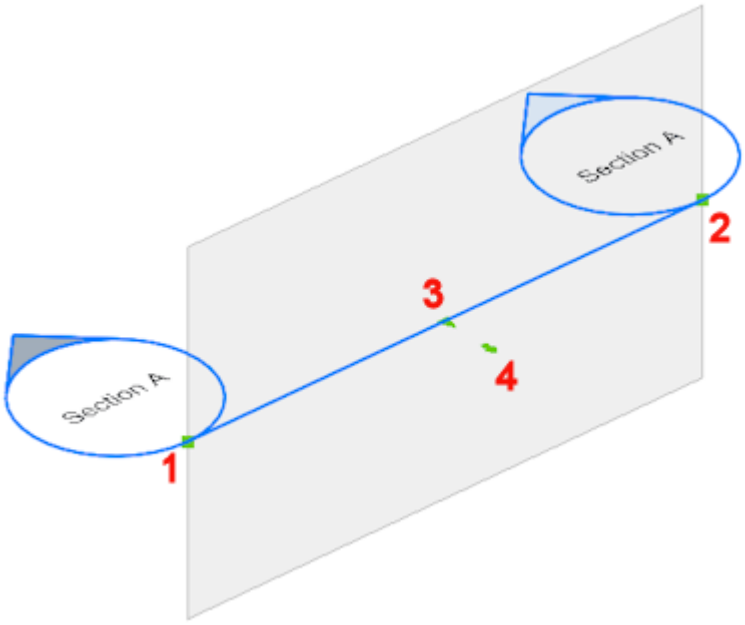
	• Use the dynamic dimension fields to define the base rectangle. Hit the TAB key to switch between the dynamic dimension fields. **Select the height of volume box:** Do one of the following: • Specify a point. • Type a value in the dynamic dimension field.
Interior	Creates interior elevations and a floor plan of a room. Prompts you: Select BIM Rooms to generate Interior Elevations: Select a room. Entities in set: 1 Select BIM Rooms to generate Interior Elevations: Select more rooms or press Enter. Interior Elevations created for 1 BIM Room(s). The *BIM / Interior Elevations* property of the selected room(s) is set to ON. Use BimSectionUpdate to update the interior elevations.

Procedures

General procedure to create a BIM section entity
Defining the BIM Section entity properties
To define a plan section
To define a cross section
To define a detail section
To create a 2D isometric drawing
To define an elevation
Turning a section plane into a BIM section entity
Creating a jogged BIM section entity

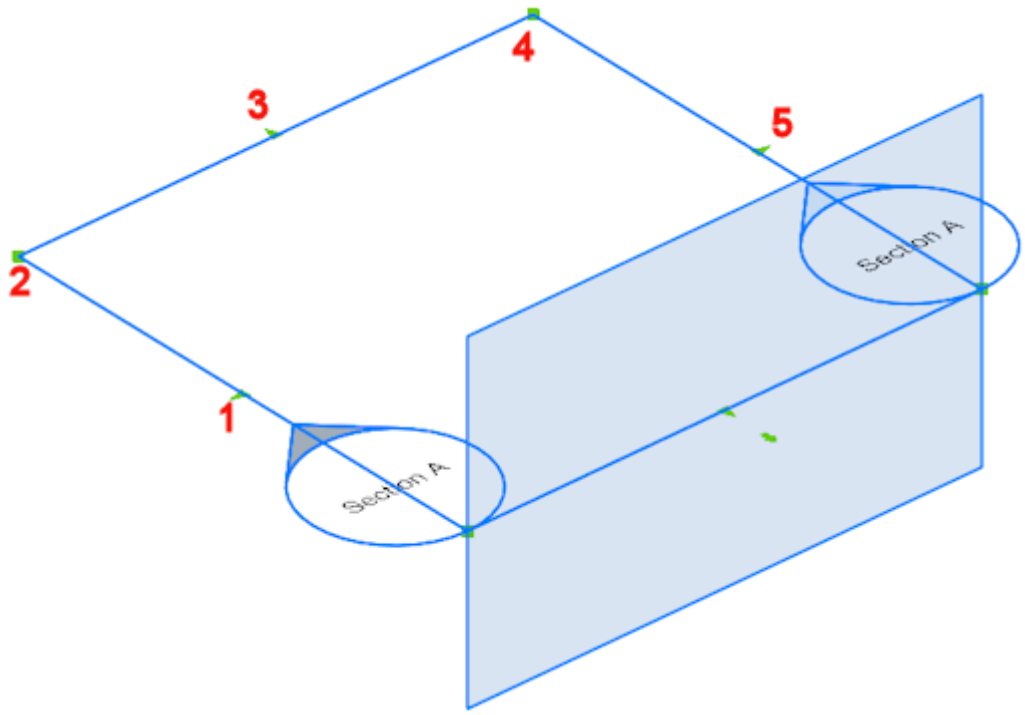
Grips Editing

BIM Section entities can be edited through grips.
Select the entity. A section entity in *Plane* state has 3 grips.



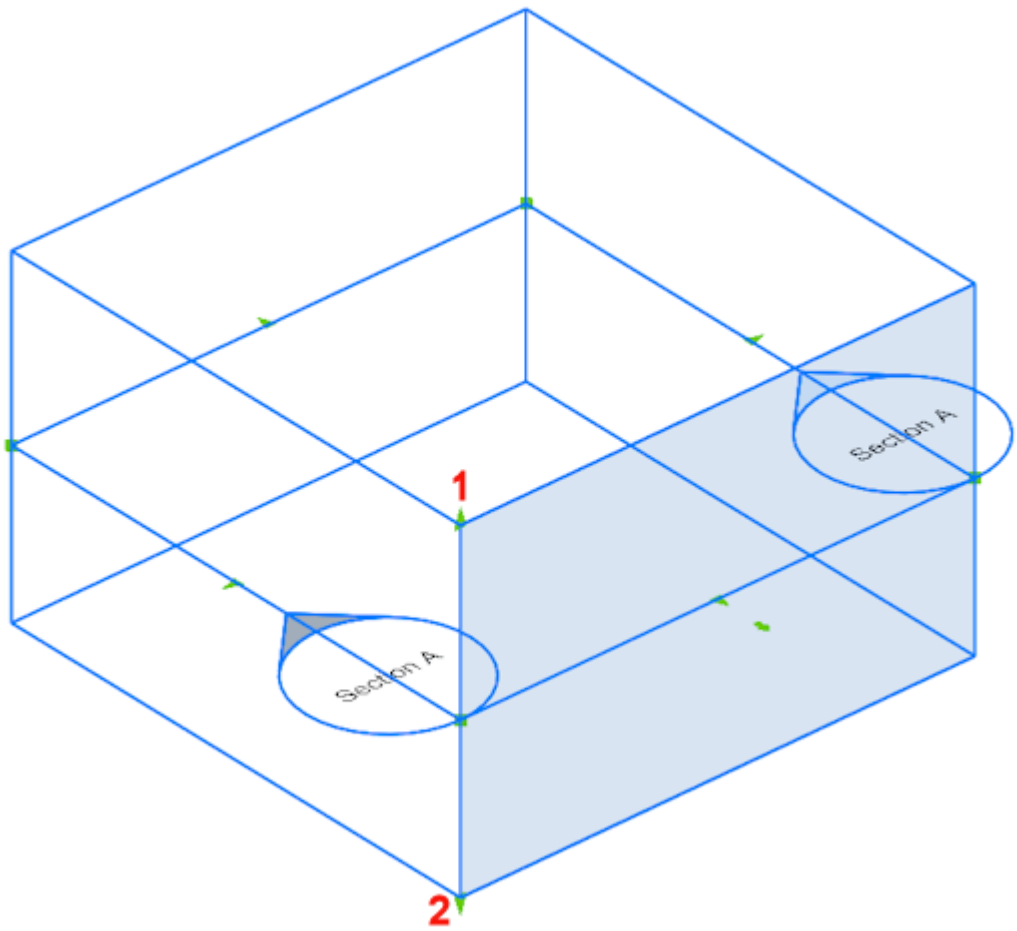
1. Start grip: allows to move the section and the position of the *Start* callout..
2. End grip: allows to modify the orientation of the section and the position of the *End* callout.
3. Center Grip: allows to move the section and the position of the *Mid* callout.
4. Click the arrow to flip the view direction.

In *Boundary* state, section entities have five more grips.



1. Midpoint grip: allows to move the section entity.
2. Corner grip: allows to deform the section boundary/volume.
3. Midpoint grip: allows to stretch the section boundary/volume.
4. Corner grip: allows to deform the section boundary/volume.
5. Midpoint grip: allows to stretch the section boundary/volume.

In *Volume* state, section entities have two more grips, compared to the boundary state..



1. Allows to move the top plane of the section entity.
2. Allows to move the bottom plane of the section entity.

Related Commands

BimPatch - allows to replace part of a generated section.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.
ClipDisplay - toggles the *Clip Display* property of a section plane or a BIM section entity.
Section - creates section planes of 3D solids; the results are region entities.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.

BimSectionOpen

Opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.

Accessing the Command

command bar: *bimsectionopen*
menu bar: *BIM | Open Section Model*
toolbar: *BIM* |
Quad: *BIM* |

: ***bimsectionopen***

To open the 3D BIM model related to a BIM section drawing

Prompts you in the command bar:

Select BIM section entity or BIM section results: (Either select a section block in Model Space or a viewport in a Paper Space layout.)

Entities in set: 1

Select BIM section entity or BIM section results: (Press Enter to open the 3D model.)

To open the drawing file(s) related to a BIM section entity

Prompts you in the command bar:

Select BIM section entity or BIM section results: (Select a BIM section entity.)

Entities in set: 1

Select BIM section entity or BIM section results: (Select more BIM section entities, or press Enter.)

Command Options

No options.

Related Commands

BimPatch - allows to replace part of a generated section.
BimSection - creates a BIM section entity.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.

BimSectionUpdate

Exports or updates the result of a BIM section in a drawing.

Accessing the Command

command bar: *bimsectionupdate*
menu bar: *BIM Update Section*
toolbar: *BIM |*

: ***bimsectionupdate***

To create or update BIM sections when in a BIM 3D model

Open the BIM 3D model.

Launch the BimSectionUpdate command.

Prompts you in the command bar:

Select BIM section entity to generate or BIM section result to update: (Select a BIM section entity or a BIM room*.)

Entities in set:1

Select BIM section entity to generate or BIM section result to update: (Select more BIM section entities or BIM rooms, or press Enter.)

Bricsys reports:

xx section(s) were generated

C:\BricsCAD\Villa\Section Plane (1)

...

C:\BricsCAD\Villa\Section Plane (xx)

*BIM rooms can be selected if the interior elevations have been created by the Interior option of the BimSection command.

To update a BIM section drawing

Open the BIM section drawing.

Launch the BimSectionUpdate command.

Prompts you in the command bar:

Select BIM section entity to generate or BIM section result to update: (Either select a section in Model Space or a viewport in a Paper Space layout.)

Entities in set:1

Select BIM section entity to generate or BIM section result to update: (Either select a section in Model Space or a viewport in a Paper Space layout; or Press Enter.)

Command Options

No Options

Related Commands

BimPatch - allows to replace part of a generated section.
BimSection - creates a BIM section entity.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.

SectionPlaneToBlock - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

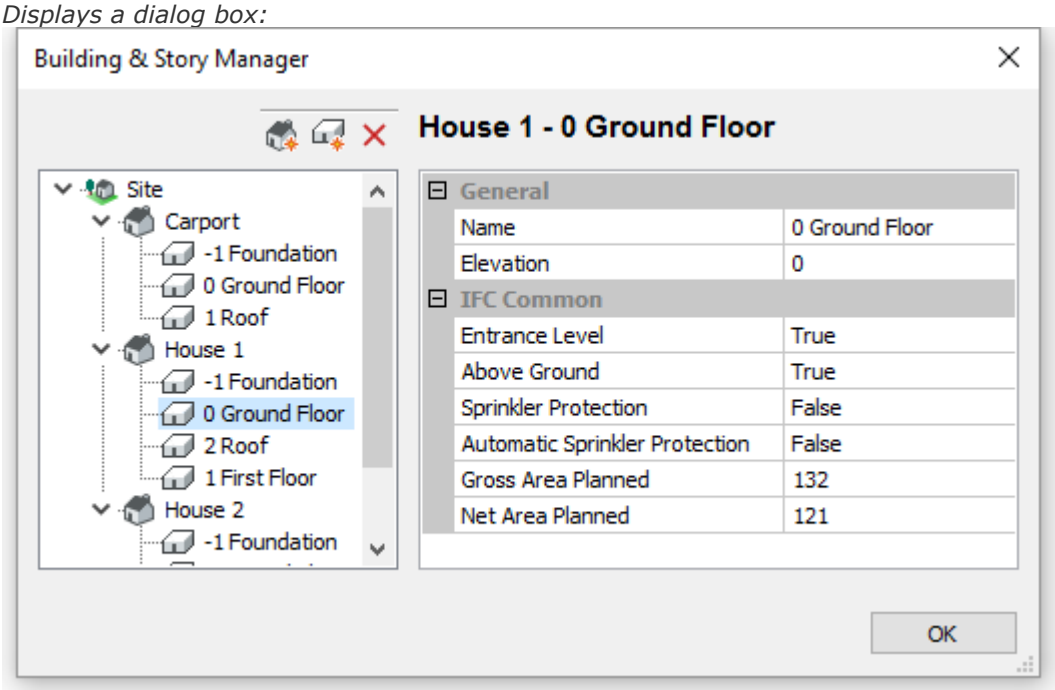
BimSpatialLocations

Opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.

Accessing the Command

command bar: *bimspatiallocations*
menu bar: *BIM | Spatial Locations*
toolbar: *BIM |*

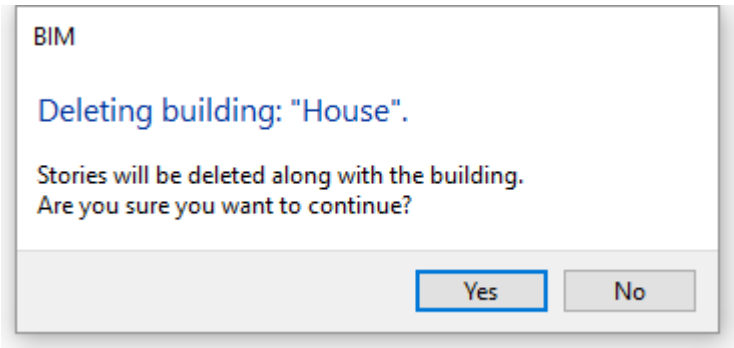
: ***bimspatiallocations***
Displays a dialog box:



Click the *Close* (X) button or the *OK* button to close the dialog box.

Command Options

<table>
<tr><th>Icon</th><th>Option</th><th>Description</th></tr>
<tr><td></td><td>New Building</td><td>Adds a new building to the model.
Fill out the properties grid:
<table>
<tr><td colspan="2">General</td></tr>
<tr><td>Name</td><td>New Building</td></tr>
<tr><td>Ref Height Elevation</td><td>0</td></tr>
<tr><td>Terrain Elevation</td><td>0</td></tr>
<tr><td colspan="2">Building Address</td></tr>
<tr><td>Internal Location</td><td></td></tr>
<tr><td>Address Lines</td><td></td></tr>
<tr><td>Postal Box</td><td></td></tr>
<tr><td>Town</td><td></td></tr>
<tr><td>Region</td><td></td></tr>
<tr><td>Postal Code</td><td></td></tr>
<tr><td>Country</td><td></td></tr>
<tr><td colspan="2">IFC Common</td></tr>
<tr><td>Building ID</td><td></td></tr>
<tr><td>Permanent ID</td><td>False</td></tr>
<tr><td>Main Fire Use</td><td></td></tr>
<tr><td>Ancillary Fire Use</td><td></td></tr>
<tr><td>Sprinkler Protection</td><td>False</td></tr>
<tr><td>Automatic Sprinkler Protection</td><td>False</td></tr>
<tr><td>Occupancy Type</td><td></td></tr>
<tr><td>Gross Planned Area</td><td>0</td></tr>
<tr><td>Number of Stories</td><td>1</td></tr>
<tr><td>Year of Construction</td><td></td></tr>
<tr><td>Land Marked</td><td>False</td></tr>
</table></td></tr>
<tr><td></td><td>New Story</td><td>Adds a new story in the selected building.
Fill out the properties grid:
<table>
<tr><td colspan="2">General</td></tr>
<tr><td>Name</td><td>New Story</td></tr>
<tr><td>Elevation</td><td>0</td></tr>
<tr><td colspan="2">IFC Common</td></tr>
<tr><td>Entrance Level</td><td>False</td></tr>
<tr><td>Above Ground</td><td>True</td></tr>
<tr><td>Sprinkler Protection</td><td>False</td></tr>
<tr><td>Automatic Sprinkler Protection</td><td>False</td></tr>
<tr><td>Gross Area Planned</td><td>0</td></tr>
<tr><td>Net Area Planned</td><td>0</td></tr>
</table></td></tr>
</table>

✕	Delete	Deletes the selected building or story. An alert box displays:  Click the *Yes* button to delete the selected item.

Procedures

Assigning a building and story to a building element
To examine the model in the Structure panel

Related Commands

BimAttachSpatialLocation - allows to attach or remove a spatial location to a selection of entities.
BimClassify - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).
Bimify - analyzes the model and runs automatic classification and spatial location assignment for the entire model.
BimRoom - creates Room entities.

BimSplit

Automatically splits segmented solids into separated solids and allows to split a solid using a selection of cutting faces.

Accessing the Command

command bar: *bimsplit*
menu bar: *Bim | Split*
toolbar: *Bim |*

: ***bimsplit***
Prompts you in the command bar:

Select solid for auto splitting or cutting faces for manual splitting: (Select a 3D solid* or one or more faces**.)

Entities in set: 1

Select solid for auto splitting or cutting faces for manual splitting: (Select more solids or faces or press Enter.)

Entity was split in 6 parts

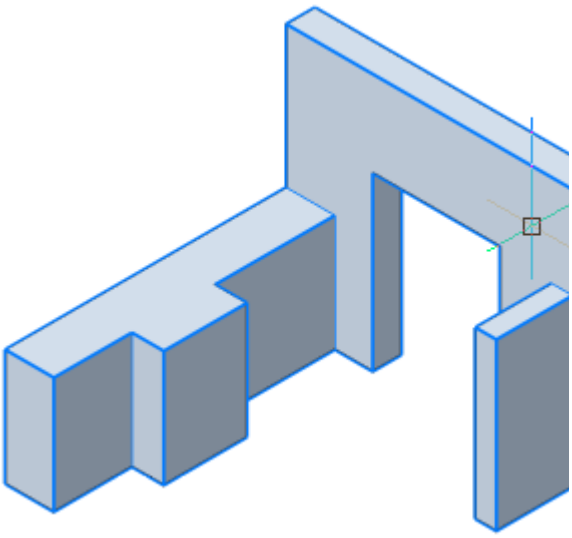
(*) To select a solid, make sure that the value of SELECTIONPREVIEW is 2 or 3; hover over the solid, then hit the TAB key until the whole solid highlights and click.

(**) To select a face, place the cursor over a face and click. Faces highlight one by one, regardless of the value of SELECTIONPREVIEW or SELECTIONMODES.

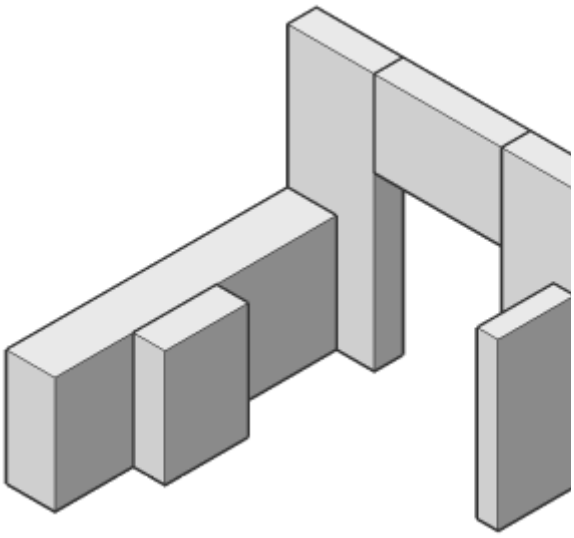
NOTE Inserts, such as windows and doors, which existed in the split solid are removed.

Command Options

Select solid:

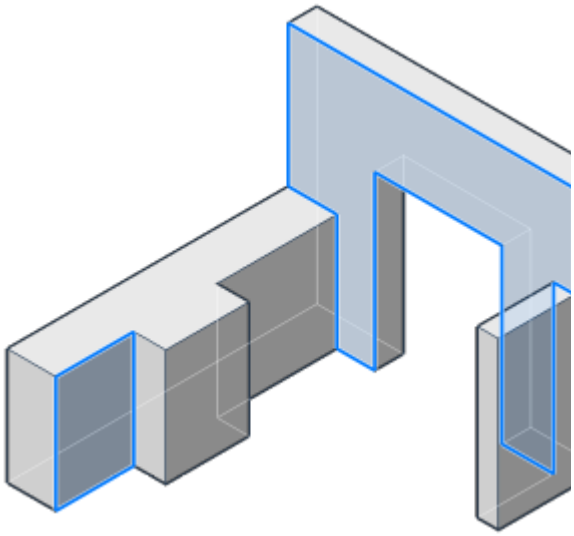


Solid selected ...

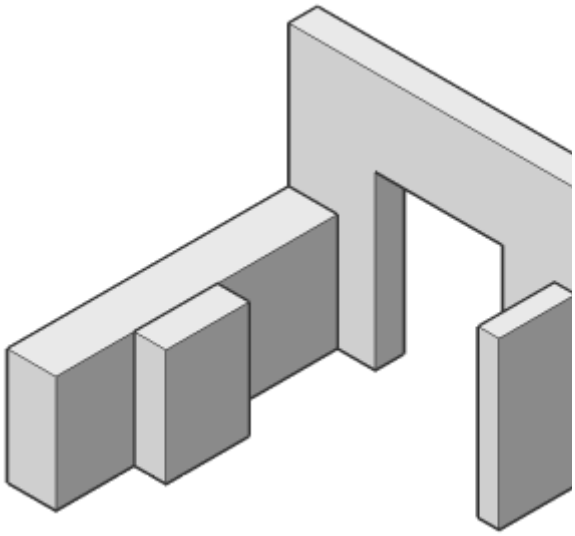


... split into 6 separate solids

Select cutting faces:



Two cutting faces selected ...



... split into 4 separate solids

Procedures

To split a solid automatically
To split a solid using cutting faces

Related Commands

BimConnect - creates an L-connection between minor faces of two solids.
Union - performs Boolean union operations on 3D solids and 2D regions; adds one to another to form a single entity.
SolidEdit - edits 3D solids and 2D regions.

BimSuggest

Analyzes the connection between solids, then finds and presents similar situations in the model where the same connection can be applied.

Accessing the Command

command bar: *bimsuggest*
Quad: *Model |*

: ***bimsuggest***
Prompts you in the command bar:

Select solids with desired detail: *(Select the first solid*.)*

Entities in set: 1

Select solids with desired detail: (Select the second solid, then right click.)

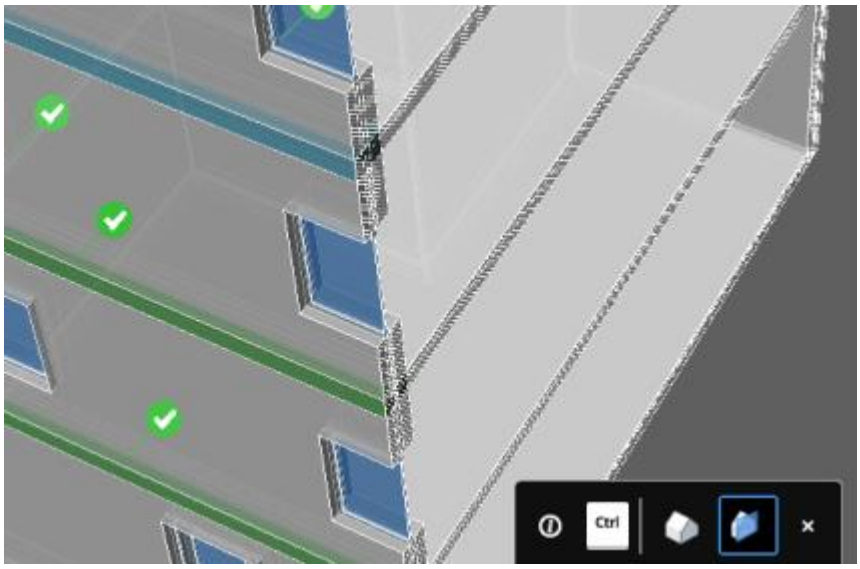
Choose next action [Apply all/Cancel] <Apply all>: (Press Enter to apply the connection to all similar situations.)

*Selection of the solids is best done in a clipped section view.

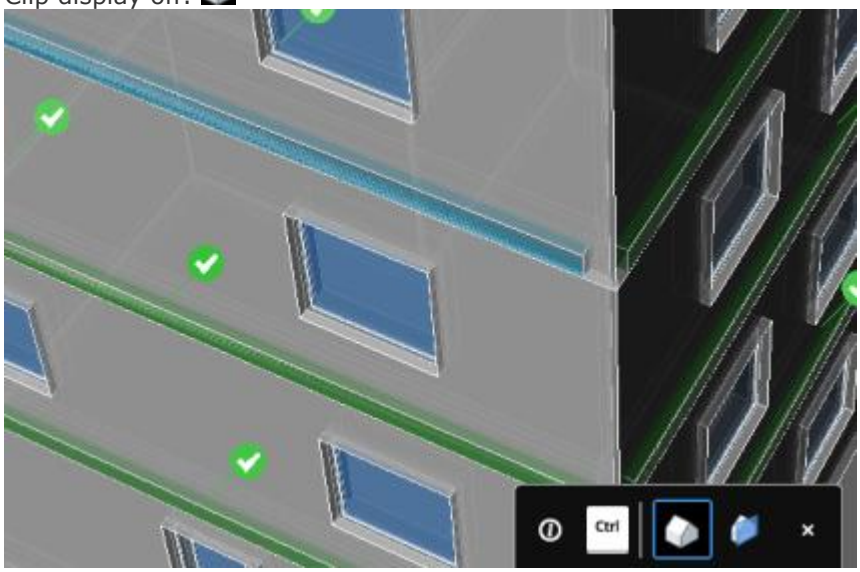
Command Options

Option	Description
Apply all	Apply the connection to all detected similar situations in the entire model.
Select or Discard	Click the selection icon to toggle the selection on/off: • : Selected location. • : Discarded location.
Toggle clip display	Allows to toggle the clip display property of the section plane(s) which where on when selectin the solids with the desired detail. Hit the Ctrl-key to toggle the clip display. If the Display tips for BIMSUGGEST options is selected the current mode is indicated in the Tips widget at the bottom of the graphic screen.

- Clip display on:



- Clip display off:



Related Commands

BimAttachComposition - attaches a BIM composition to solids.
BimDrag - drags minor faces, optionally connects minor faces to major faces of other solids.
BimSection - creates a BIM section entity.

BimTag

Creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.
The command can be used on section viewports only.

Accessing the Command

command bar: *bimtag*
ribbon: *Annotate | Tag*

: ***bimtag***
Prompts you in the command bar:
Select BIM section viewport [Manual]: (Click the border of a Bim Section viewport.)

Tags are created automatically for all sectioned building entities in the selected viewport.

The tags are created as multileader entities using the blocks in the *_ Section Tag.dwg* in the *Bim/Sections* folder of the support folder, e.g.:
C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V18x64\en_US\Support\Bim\Sections.

If tags already exist in the selected viewport, prompts you:
Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>: (Press Enter to (re)generate all tags or choose an option.)

Command Options

Option	Description
Manual	Prompts you: **Pick point on a sectioned BIM element [Auto]:** Click a sectioned bim element Prompts you: Place the tag. Pick a point to place the tag. Prompts you: Pick point on a sectioned BIM element [Auto]: Click a sectioned bim element or press Enter to stop.
Auto	Tags are created automatically for all sectioned building entities in the selected viewport.
Update Existing	Updates the existing tags in the selected viewport.
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.

Related Commands

BimSectionUpdate - exports or updates the result of a BIM section in a drawing.
BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
BimRoom - creates BIM Room entities.
MLeader - creates multileader entities using the current multileader style.
MLeaderEdit - adds leader lines to and removes leader lines from a multileader entity.

BimUpdateRoom

Updates the geometry of a Room entity.

Accessing the Command

command bar: *bimupdateroom*
menu bar: *BIM | Update Room*
toolbar: *BIM |*

: ***bimupdateroom***
Prompts you in the command bar:
Select Room: *(Select a Room entity.)*

Command Options

No command options.

Related Commands

BimRoom - creates Room entities.

BimUpdateThickness

Re-applies the overall thickness of a composition to the solid.

Accessing the Command

command bar: *bimupdatethickness*
menu bar: *BIM | Update Thickness*
toolbar: *BIM |*

: ***bimpupdatethickness***
Prompts you in the command bar:

Select entities to update thickness:: (Select an entity.)
Entities in set: 1

Select entities to update thickness: (Select an entity or press Enter.)

Done on 1 entity(ies).
Thickness of x element(s) has been updated. Skipped: y element(s).

Command Options

No options.

Procedures

Editing a composition

Related Commands

BimAttachComposition - attaches a BIM composition to solids.
Bimflip - flips the starting face from which the layers of a composition are set out.

BimWindowCreate

Creates a fully parametric window based on a closed contour.

Accessing the Command

command bar: *bimwindowcreate*
menu bar: *Bim | Create Window*
toolbar: *Bim | Create Window*
Quad: *Model | Create Window*

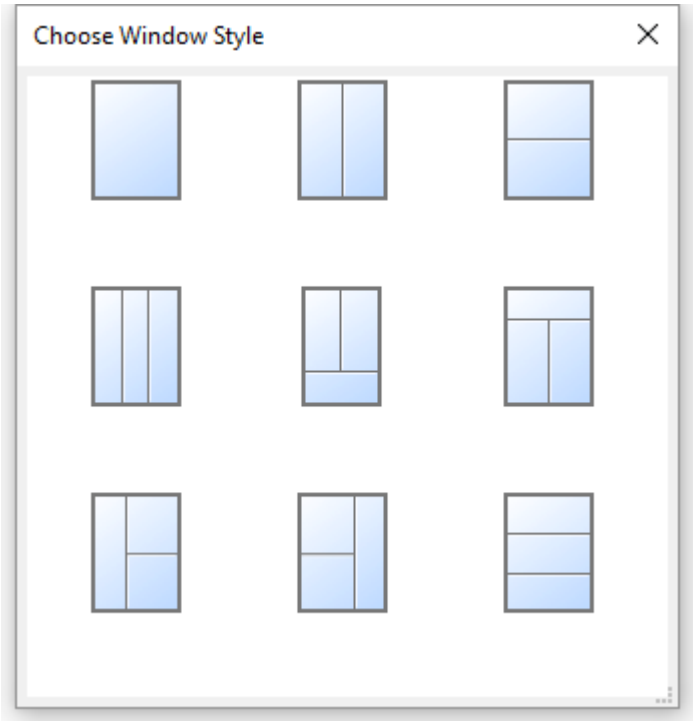
: ***bimwindowcreate***
Prompts you in the command bar:

Select the window's profile: (Select a closed 2D entity or boundary.)

Entities in set: 1

Select the window's profile: (Select a closed 2D entity or boundary, or press Enter.)

A Dialog box displays:



Pick a window style.
The window(s) is (are) created using the selected style.

Command Options

No command options.

Procedures

To create a window using a closed 2D entity
To create a window using a boundary
To create window in an existing opening

Related Commands

BimInsert - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.
BimWindowUpdate - updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

BimWindowUpdate

Updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

Accessing the Command

command bar: *bimwindowupdate*
menu bar: *BIM| BIM Update*
toolbar: *BIM |*

: ***bimwindowupdate***
Prompts you in the command bar:
Select windows to update: *(Select a window or door.)*
Entities in set: 1
Select windows to update: *(Select another window or door or right click.)*

Command Options

No options.

Related Commands

BimInsert - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.
BimWindowCreate - creates a fully parametric window based on a closed contour.

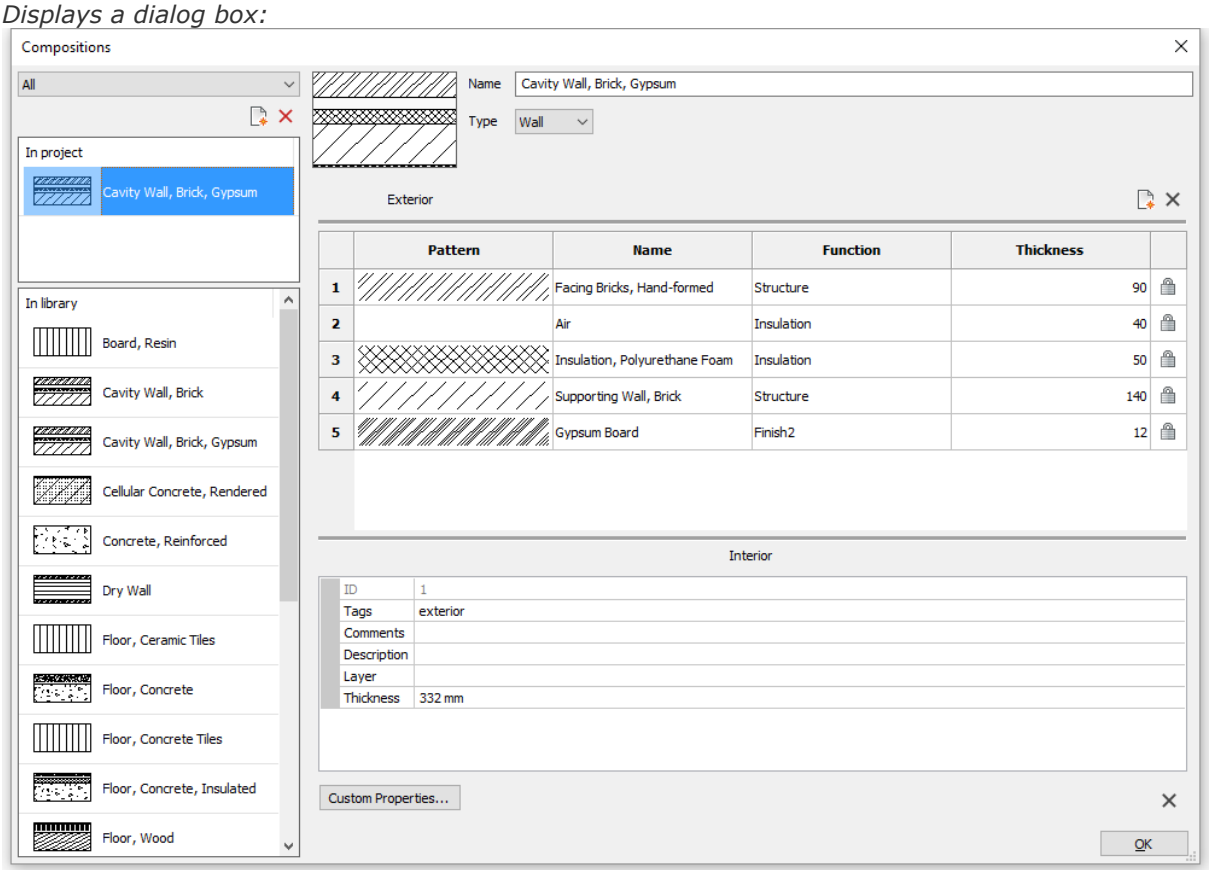
BLCompositions

Opens the compositions library in the current project and the referenced library.

Accessing the Command

command bar: *blcompositions*

: ***blcompositions***
Displays a dialog box:



Procedures

About the composition thickness
Adding compositions
Editing a composition
To copy compositions between the Project and Library databases
Deleting compositions
Attaching a composition to solids
Displaying the composition plies

Related Commands

BimAttachComposition - attaches a BIM composition to solids.
BimProfiles - opens the profiles library in the current project and the referenced library.
BimProjectInfo - opens the BIM Project Info dialog box.
BLMaterials - opens the physical materials library in the current project and the referenced library.

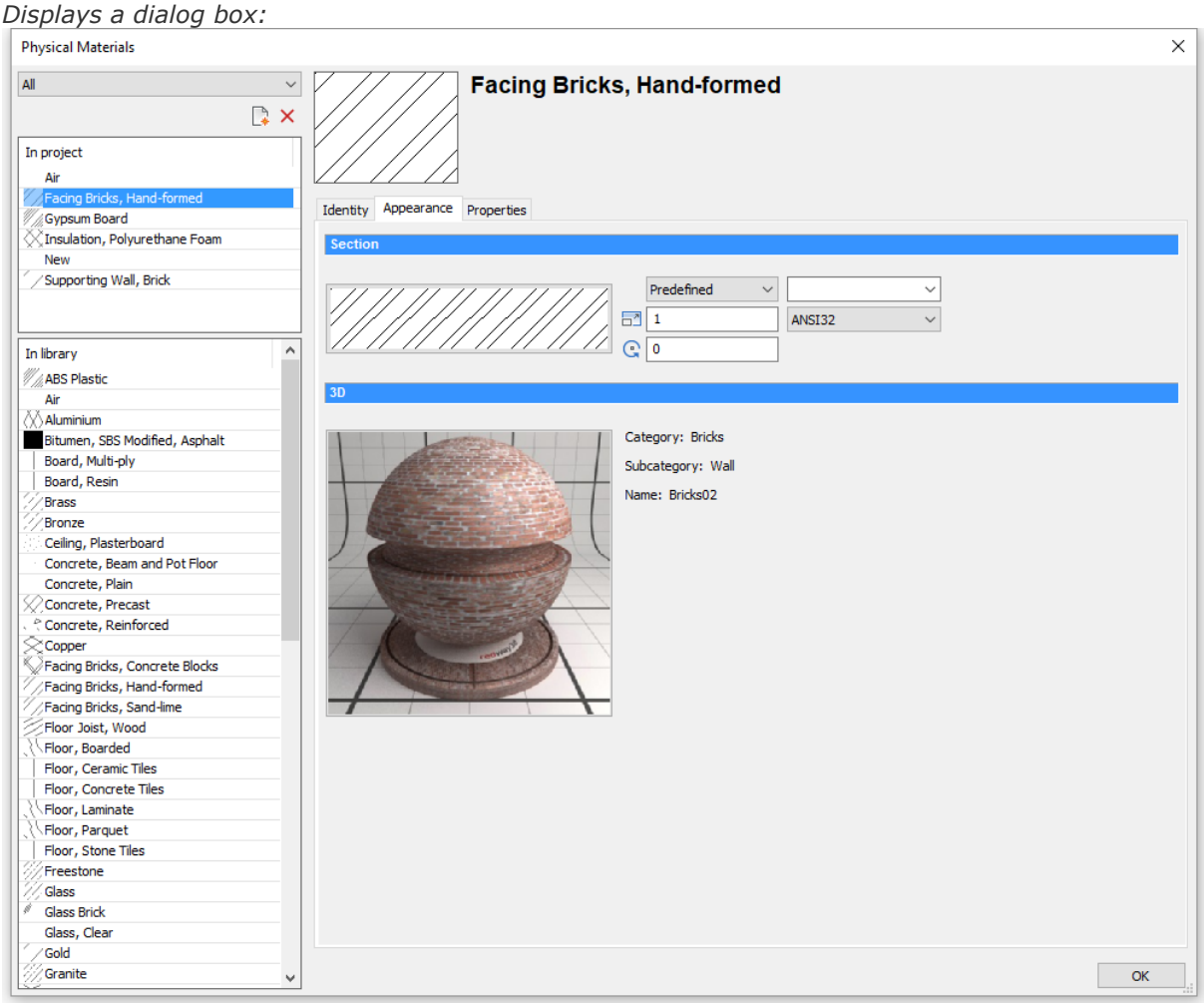
BLMaterials

Opens the Physical Materials library in the current project and the referenced library.

Accessing the Command

command bar: *blmaterials*

: ***blmaterials***
Displays a dialog box:



Procedures

Adding physical materials
Editing a physical material
To copy materials between the Project and Library databases
Deleting physical materials

Related Commands

BimProfiles - opens the profiles library in the current project and the referenced library.
BimProjectInfo - opens the BIM Project Info dialog box.

BLCompositions - opens the compositions library in the current project and the referenced library.

Bricsys 24/7

CloudAccount

Reports the current status of your Bricsys 24/7 account in the command bar.

Accessing the Command

command bar: c*loudaccount*

: ***cloudaccount***
Reports in the command bar:
If logged in:
Server: https://my.bricsys247.com
User name: your.name@email.com

If logged off:
Bricsys 24/7: not logged in.

Command Options

No options.

Related Commands

CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogoff - logs off from Bricsys 24/7.
CloudLogon - logs on to Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
CloudUpload - uploads the current drawing to Bricsys 24/7.

CloudDownload

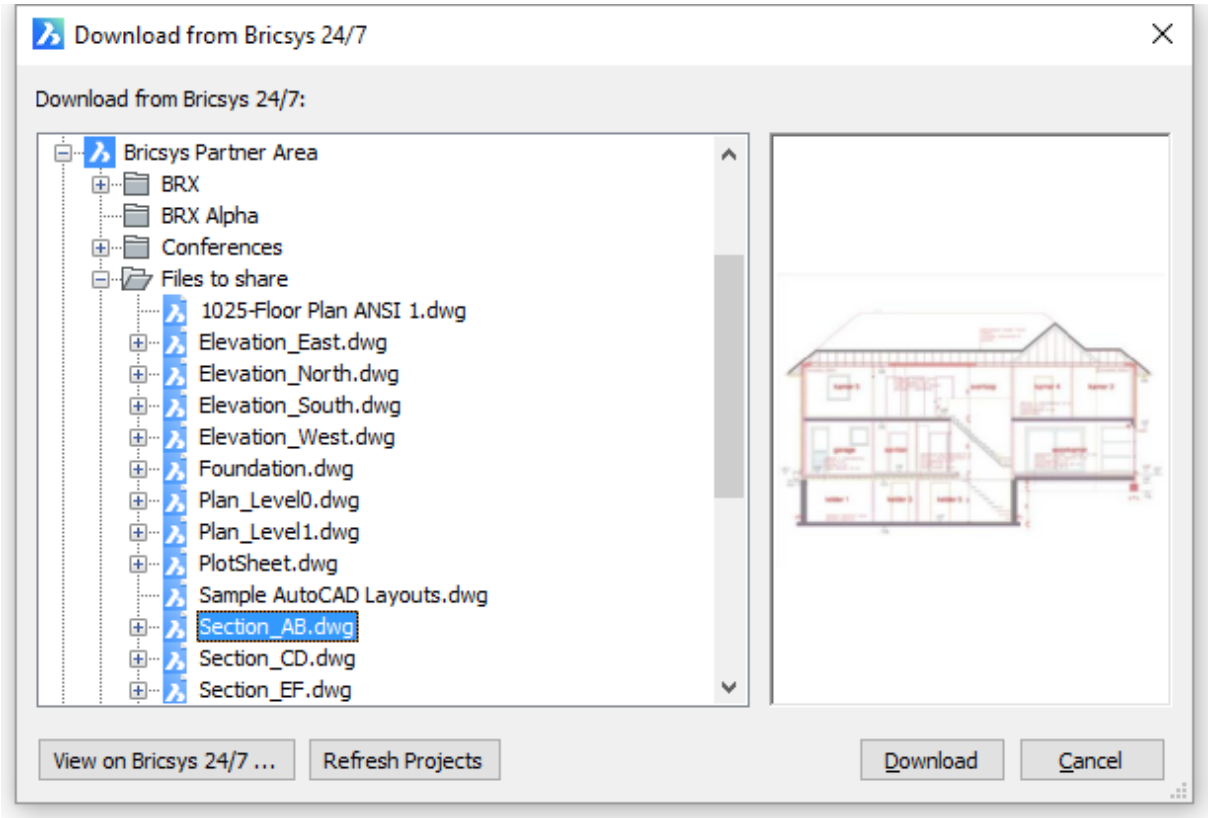
Downloads a drawing from your Bricsys 24/7 project to a local folder.

Accessing the Command

command bar: *clouddownload*
menu bar: *File | Bricsys 24/7 | Download...*
toolbar: *Bricsys 24/7 |*

: ***clouddownload***

If you are not yet logged on to Cloud, the CloudLogon command is launched first.
Displays a dialog box:



Command Options

Option	Description
View on Bricsys 24/7...	Shows the selected document in the Cloud viewer.
Refresh Projects	Reloads the project list.
Download	Downloads the selected document to a local folder.

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudLogoff - logs off from Bricsys 24/7.
CloudLogon - logs on to Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
CloudUpload - uploads the current drawing to Bricsys 24/7.

CloudLogoff

Logs off from your Bricsys 24/7 project.

Accessing the Command

command bar: *cloudlogoff*

: ***cloudlogoff***
Logged off.

Command Options

No options

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogon - logs on to Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
CloudUpload - uploads the current drawing to Bricsys 24/7.

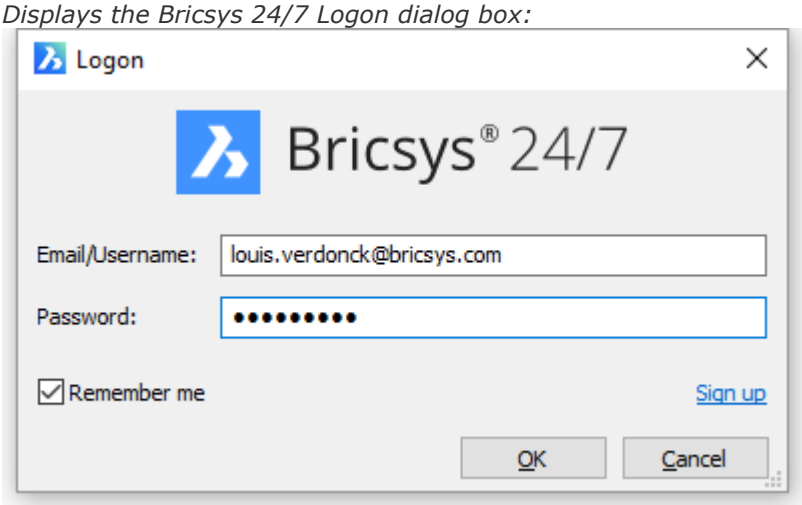
CloudLogon

Logs on to Bricsys 24/7.

Accessing the Command

command bar: *cloudlogon*

: ***command***
Displays the Bricsys 24/7 Logon dialog box:



Fill out the *E-mail/Username* and *Password* fields, then press the *OK* button.
Optionally check the *Remember me* option.

Command Options

Option	Description
Sign up	Allows to create a Bricsys 24/7 account.

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogoff - logs off from Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
CloudUpload - uploads the current drawing to Bricsys 24/7.

CloudOpen

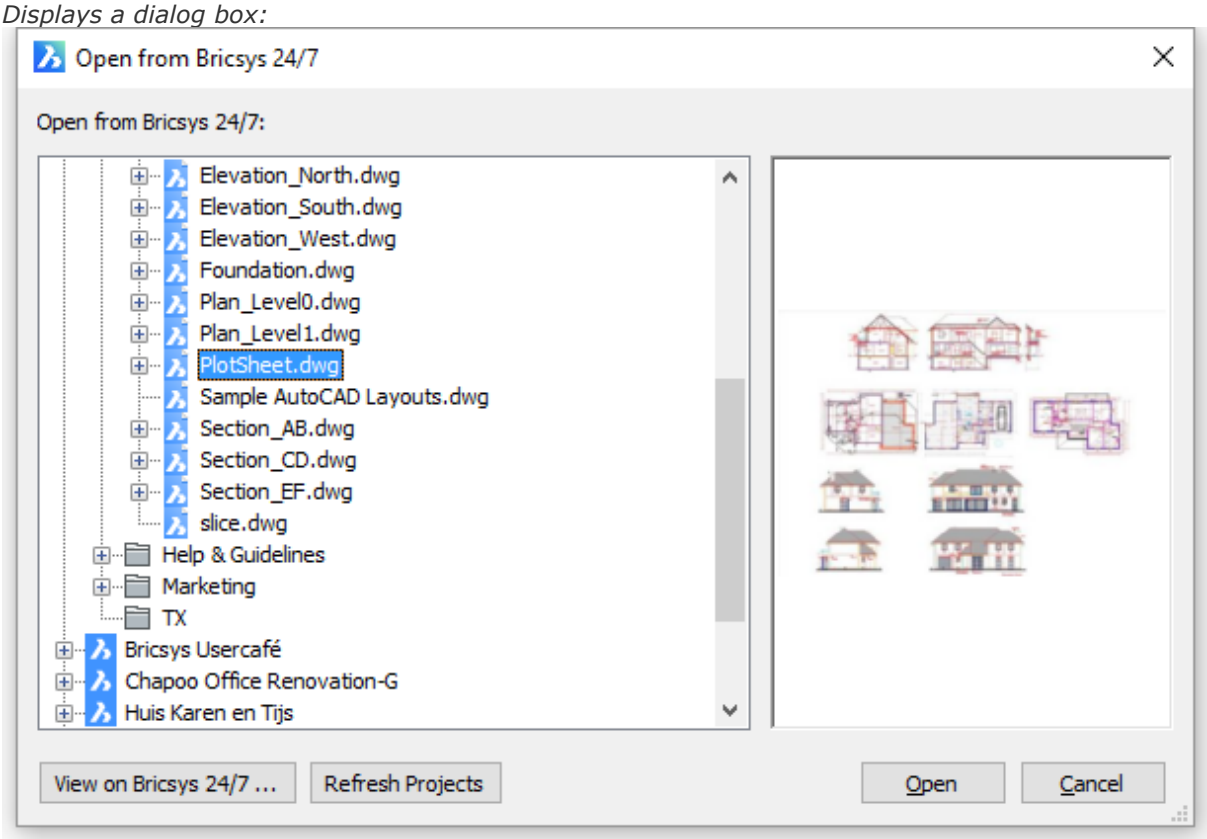
Downloads and opens a drawing from Bricsys 24/7.

Accessing the Command

command bar: *cloudopen*
menu bar: *File | Bricsys 24/7 | Open...*
toolbar: *Bricsys 24/7 |*

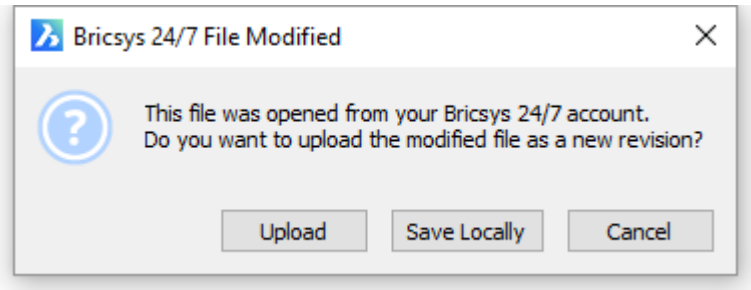
: ***cloudopen***

If you are not yet logged on to Bricsys 24/7, the CloudLogon command is launched first.
Displays a dialog box:

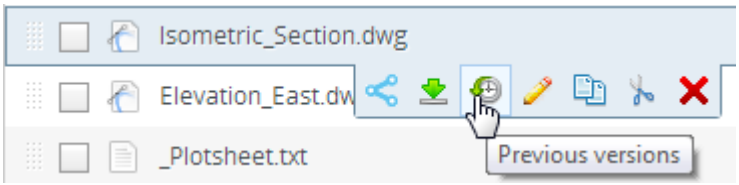


Command Options

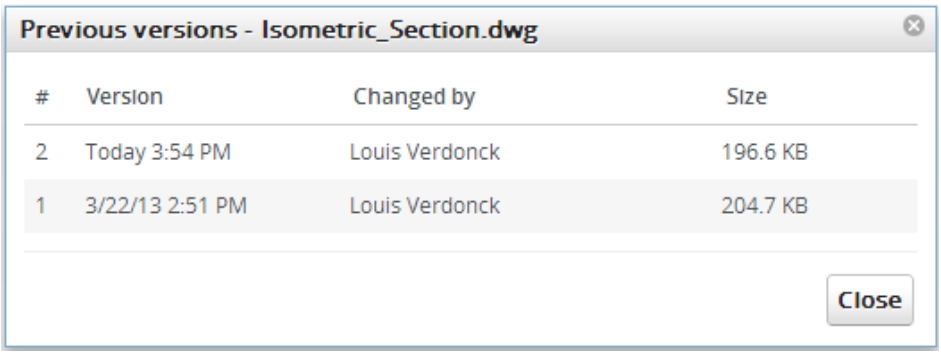
Option	Description
View on Bricsys 24/7...	Shows the selected document in the Cloud viewer.
Open	Downloads and opens the selected drawing. A copy of the drawing is saved in the subfolder of the folder which is defined in the CloudTempFolder user preference (e.g. C:\Users\<UserName>\AppData\Local\Temp\Bricsys_24_7\). When saving the drawing, a dialog displays:



- Upload: Uploads the drawing to Bricsys 24/7 as a revision of the original drawing.
  On Bricsys 24/7 the *Previous Versions* icon ( ) displays when hovering over a drawing of which revisions have been uploaded.

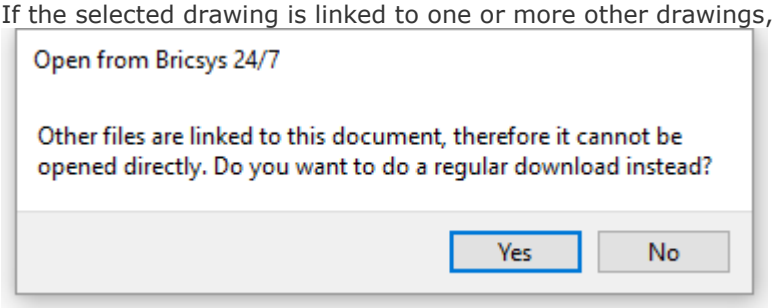


  Click the *Previous Versions* icon ( ) to see a list of all available revisions:



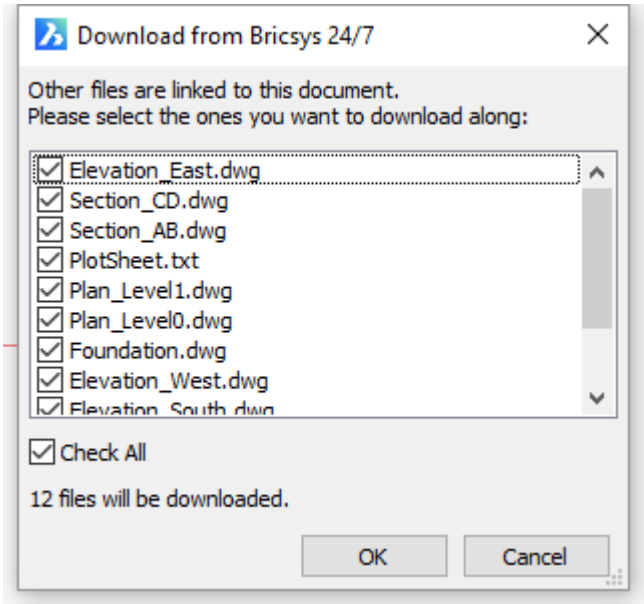
- Save Locally: Saves a copy of the drawing locally.

If the selected drawing is linked to one or more other drawings, a dialog displays:



Do one of the following:

- Click the *Yes* button and select the linked documents you want to download.



- Click the *No* button to conclude the command. The drawing is not downloaded.

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogoff - logs off from Bricsys 24/7.
CloudLogon - logs on to Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
CloudUpload - uploads the current drawing to Bricsys 24/7.

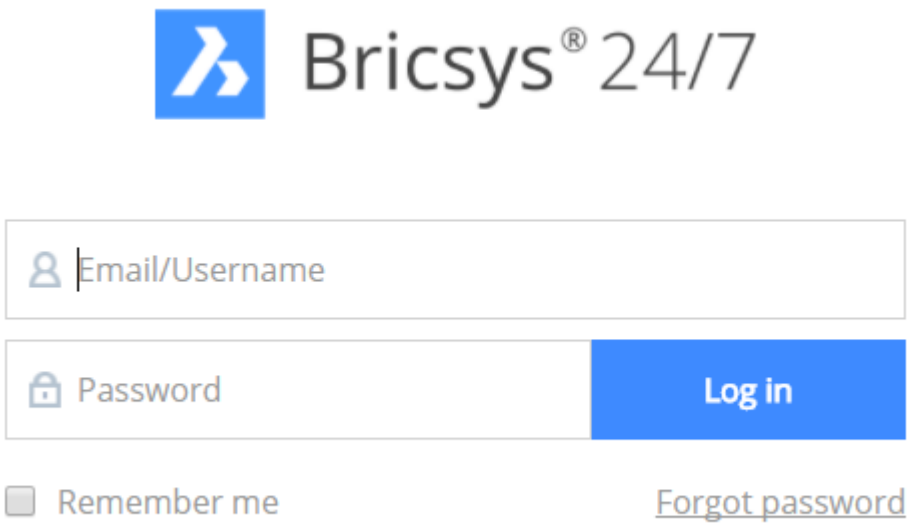
CloudProject

Opens your Bricsys 24/7 account in a browser.

Accessing the Command

command bar: *cloudproject*
menu bar: *Bricsys 24/7 | My Bricsys 24/7...*
toolbar: *Bricsys 24/7|*

: ***cloudproject***
Displays the logon page to Bricsys 24/7in a browser:



Command Options

No command options.

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogoff - logs off from Bricsys 24/7.
CloudLogon - logs on to Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudUpload - uploads the current drawing to Bricsys 24/7.

CloudUpload

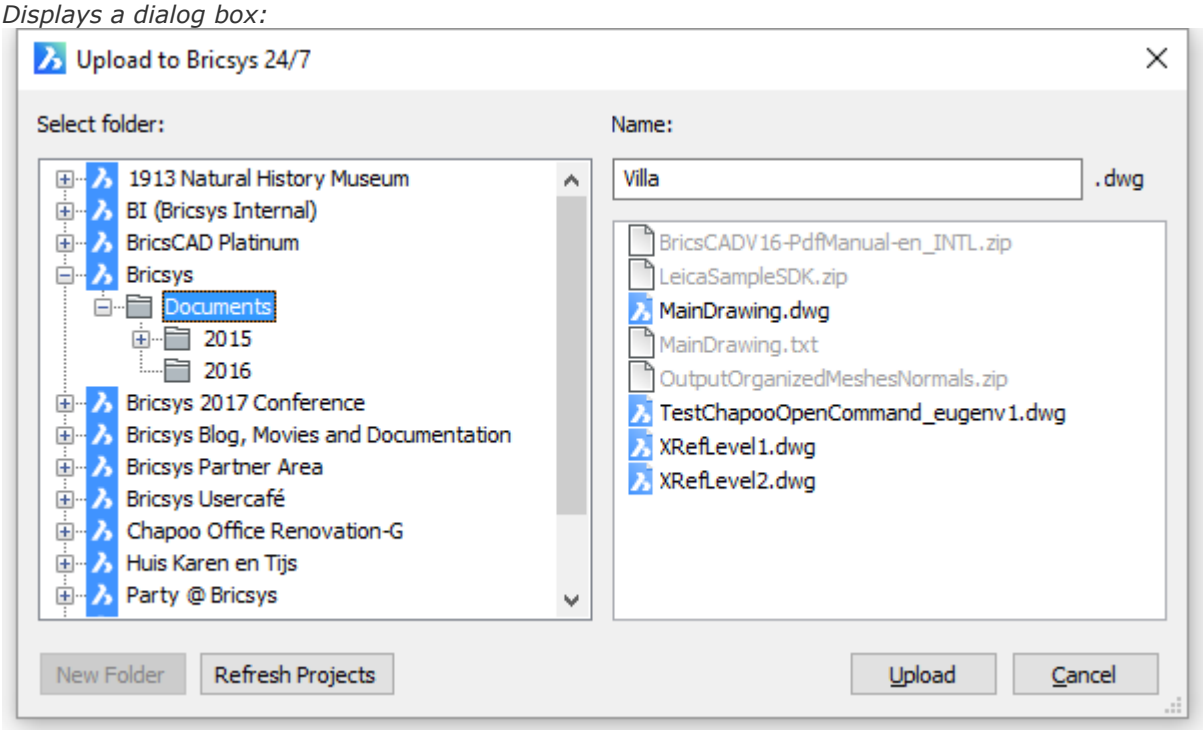
Uploads the current drawing to Bricsys 24/7.

Accessing the Command

command bar: *cloudupload*
menu bar: *File | Bricsys 24/7 | Upload...*
toolbar: *Bricsys 24/7 |*

: ***cloudupload***

If you are not yet logged on to Bricsys 24/7, the CloudLogon command is launched first.
Displays a dialog box:

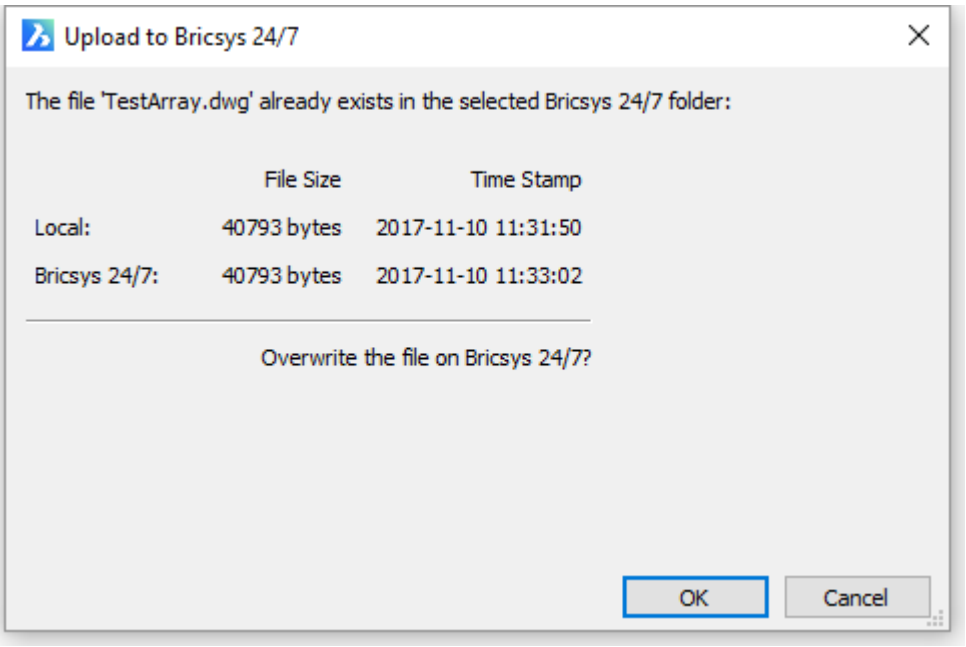


Expand a project, then select a folder.

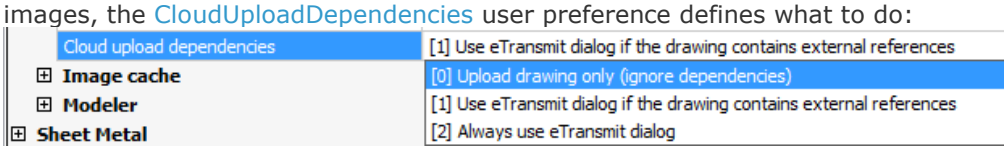
Click the Upload button.

Command Options

Option	Description
New Folder	Creates a new folder as a subfolder of the currently selected folder.
Refresh Projects	Reloads the project list.
Upload	Uploads the drawing to the selected folder. If a drawing of the same name already exists in the selected folder, the drawing is uploaded as a revision of the original drawing.



If the drawing is dependent from other files, such as external references or images, the CloudUploadDependencies user preference defines what to do:



- 0: Upload the drawing only and ignore dependencies.
- 1: Launch the eTransmit command first to compose a zip-file if the drawing contains external references.
- 2: Always launch eTransmit first.

Related Commands

CloudAccount - reports the current status of your Bricsys 24/7 account in the command bar.
CloudDownload - downloads a drawing from your Bricsys 24/7 project to a local folder.
CloudLogoff - logs off from Bricsys 24/7.
CloudLogon - logs on to Bricsys 24/7.
CloudOpen - downloads and opens a drawing from Bricsys 24/7.
CloudProject - opens your Bricsys 24/7 project in a browser.
Etransmit - creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

Chapoo

Chapoo commands are obsolete. See Bricsys 24/7 cloud commands instead.

C

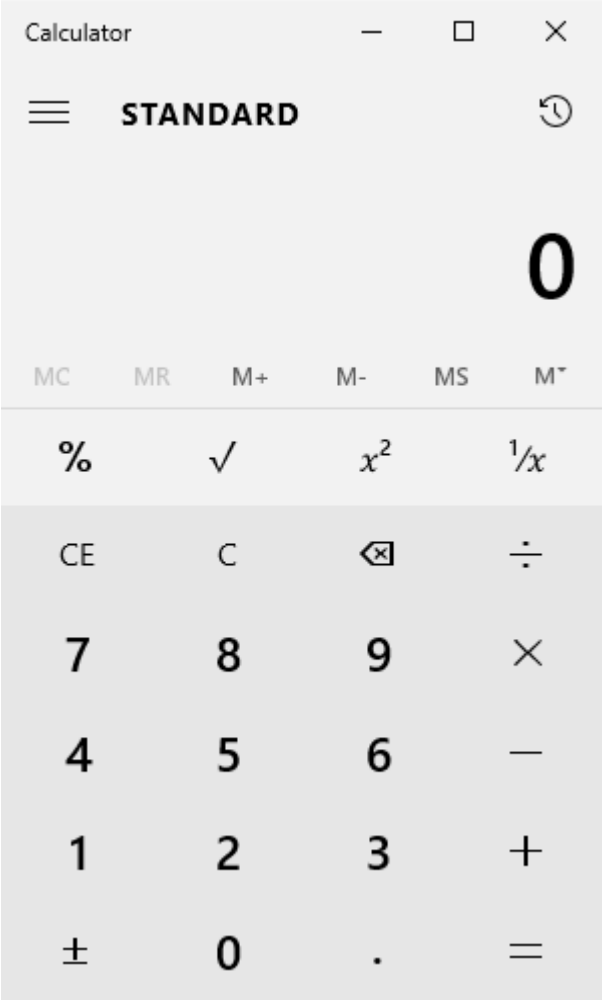
Cal

Displays the operating system Calculator.

Accessing the Command

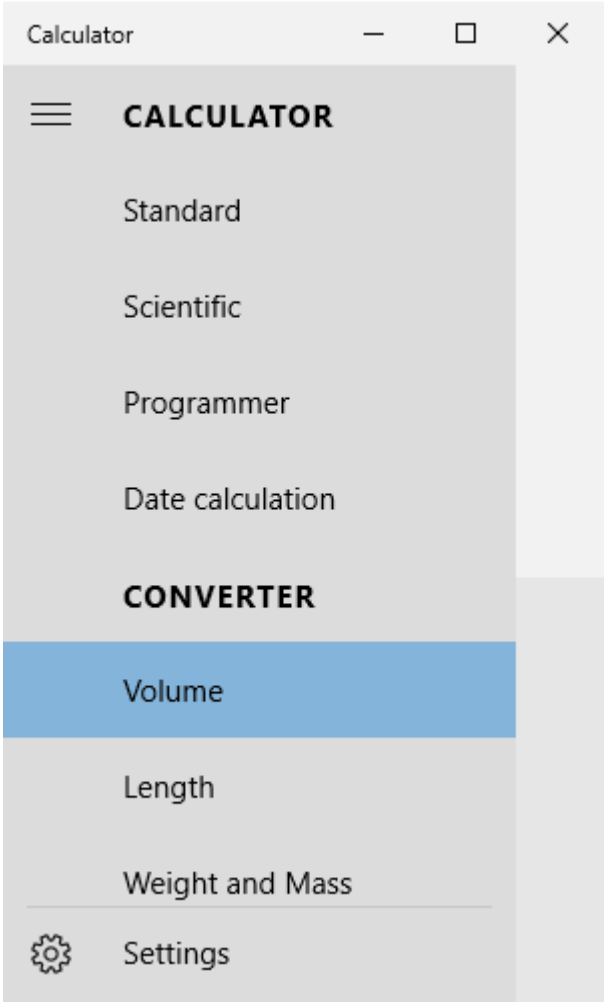
command bar: *cal*
transparent: *'cal*

: ***Cal***
Displays a dialog box (Windows 10):



Enter numbers and operations, and then press =.
For scientific calculations, choose View | Scientific.
To close the calculator, click X.

Command Options

Option	Description
View	 Click the *Mode* button (☰) and choose an option.

Related Commands

Shell - access the command prompt of the operating system; runs Windows programs.
Files - displays the operating system's File Manager.

Camera

Creates perspective views.

Accessing the Command

command bar: *camera*
menu bar: *View | Camera*
toolbar: *View |*

: ***camera***
Prompts you in the command bar:

Specify camera location: **(Enter a point to specify the location of the camera)**
Specify target location: (Enter a point to specify the location of the target)

Enter an option [?/Name/LOcation/Height/Target/LEns/CLipping/View/eXit] <eXit>: **(Enter an option or press Enter to conclude the command**

A Camera glyph displays in the drawing to indicate the position of the camera.

A saved view is created in the drawing.

Command Options

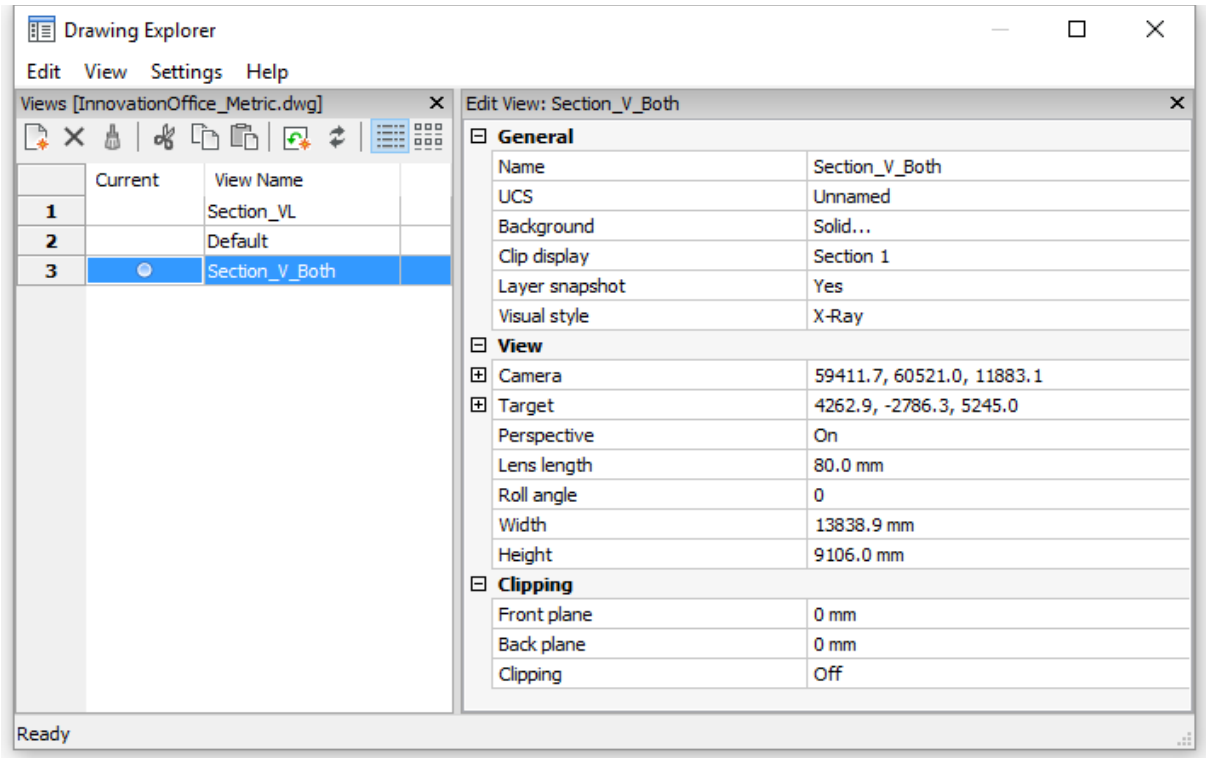
Option	Description
?	Displays a list of existing cameras. Prompts you: Enter camera name(s) to list <*>: Do one of the following: • Press Enter to list all existing cameras. • Use wildcards to (* or ?) list a selection of cameras. Cam* lists all cameras which name start with 'cam' ?a* lists all camera names of which the second letter is 'a'
Name	Prompts you: Enter name for new camera <Camera1>: Type a name for the saved view or press Enter to accept the default name.
Location	Repositions the camera. Prompts you: Specify camera location <x,y,z>: Do one of the following: • Pick a point in the drawing. • Type the x-, y- and z-coordinates in the command bar, then press Enter.
Height	Sets the Z-coordinate of the camera location. Prompts you: Specify camera height <current height>: Enter a value.
Target	Repositions the target. Prompts you: Specify target location <x,y,z>: Do one of the following: • Pick a point in the drawing. • Type the x-, y- and z-coordinates in the command bar, then press Enter.
Lens	Defines the lens length. Prompts you: Specify lens length in mm <50.00>: Enter a value.
Clipping	Defines the front and back clipping planes. Prompts you:

	Enable front clipping plane? Yes/<No>: Type *Y* to define the front clipping plane. Prompts you: Specify front clipping plane offset from target plane <0.00>: Do one of the following: • Pick to point in the drawing. • Type a value in the command bar, then press Enter Enable back clipping plane? Yes/<No>: Type *Y* to define the back clipping plane. Prompts you: Specify back clipping plane offset from target plane <0.00>: Do one of the following: • Pick to point in the drawing. • Type a value in the command bar, then press Enter
View	Sets the camera view current. Prompts you: Switch to camera view? Yes/<No>: Type *Y*, then press Enter.
Exit	Concludes the command and creates the camera.

Editing a Camera

Properties of a camera view can be edited:

- In the Drawing Explorer - Views dialog.



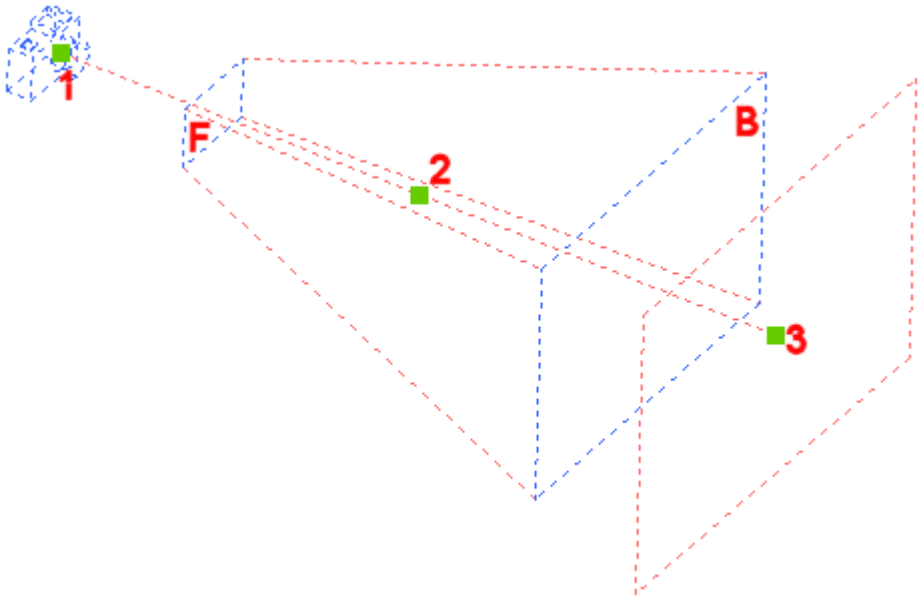
- In the Properties bar: Select the camera glyph in the drawing.

Camera	
Name	Camera1
Camera	-422.40, -2318.27, 0.00
X	-422.40
Y	-2318.27
Z	0.00
Target	5551.65, -2396.75, 0.00
X	5551.65
Y	-2396.75
Z	0.00
Lens Length	50.00
Field of view	38°34'48"
Roll angle	0°0'0"
Plot glyph	No
Clipping	
Front plane	4500.00
Back plane	1000.00
Clipping	Front and back on

Grips Editing

Cameras can be edited directly through grips:

1. Select the camera glyph in the drawing. Notice that it has three grips. Front (F) and back (B) clipping planes display in a blue dotted line.
2. Drag the grips to perform the following actions:
   - Position the camera (1).
   - Move the camera definition as a whole (2).
   - Position the target (3).



Related Commands

AniPath - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
DView - changes the 3D viewpoint interactively, and turns on perspective mode.
MatchPerspective - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs.
View - creates and sets named views through the Drawing Explorer.

Center

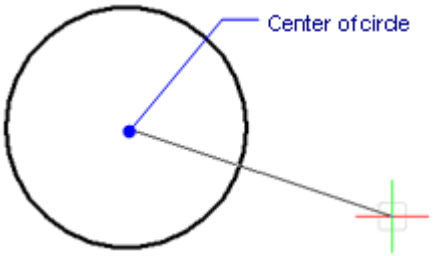
Toggles Center entity snap; snaps to the center of circles, arcs, and other entities with a circular curve.

Accessing the Command

command bar: *center*
transparent: *cen*
toolbar: *Entity Snaps* |

: ***center***

Displays no prompts in the command bar.



When center entity snap is on, the cursor snaps to the center points of curved entities, such as circles, arcs, and polyarcs.

Command Options

Option	Description
Center	Entering the command toggles Center entity snap. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.

Tangent - toggles tangent snap mode.

CenterDisassociate

Breaks the association of centerline and a center mark entities.

Accessing the Command

command bar: *centerdiassociate*

: ***centerdiassociate***
Prompts you in the command bar:

Select a center mark or centerline to disassociate: (Select a center mark or centerline entity.)
Entities in set: 1

Select a center mark or centerline to disassociate: (Select a center mark or centerline entity, or press Enter.)

Command Options

No command options.

Related Commands

CenterLine - creates a centerline entity associated with two selected lines.
CenterMark - creates a center mark entity associated with a selected circle or arc.
CenterReassociate - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.
CenterReset - resets centerline and center mark entities.

CenterLine

Creates a centerline entity associated with two selected lines.

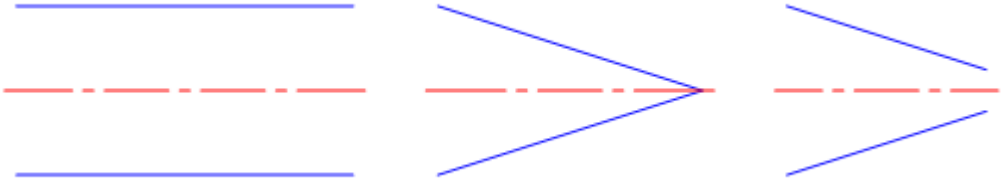
Accessing the Command

command bar: *centerline*

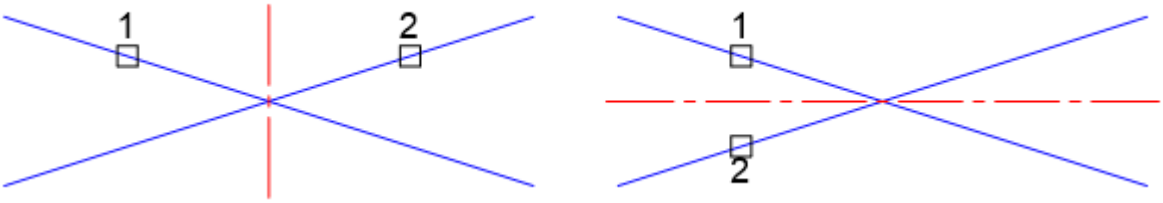
: ***centerline***
Prompts you in the command bar:

Select first line: (Select a line entity.)

Select second line: (Select a line entity.)



When creating a centerline of two intersecting lines, the location of the picked points define the direction of the centerline.



Command Options

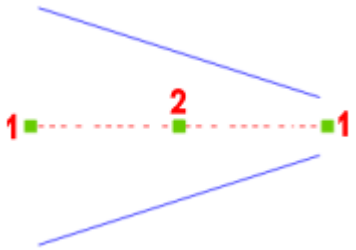
No command options.

Procedures

Defining system variables for center entities

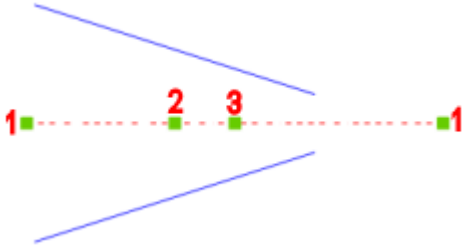
Grips Editing

Centerlines can be edited through grips.



1. Overshoot grips: drag the grip to lengthen the centerline
2. Midpoint grip: drag the grip to move the centerline*.

When an extension is applied to one or both ends of the centerline by dragging an overshoot grip, four grips display.



1. Overshoot grips: drag the grip to lengthen the centerline
2. Midpoint of the original centerline; drag the grip to move the centerline*.
3. Midpoint of the extended centerline; drag the grip to move the centerline*.

*Use the CenterReset command to reset the centerline.

Related Commands

CenterDisassociate - breaks the association of centerline and a center mark entities.
CenterMark - creates a center mark entity associated with a selected circle or arc.
CenterReassociate - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.
CenterReset - resets centerline and center mark entities.

CenterMark

Creates a center mark entity associated with a selected circle or arc.

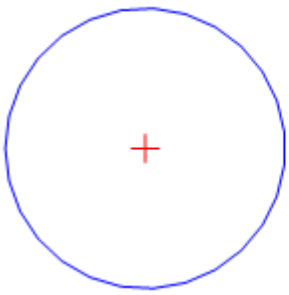
Accessing the Command

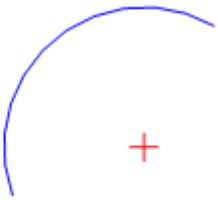
command bar: *centermark*

: ***centermark***
Prompts you in the command bar:

Select circle or arc: (Select a circle or a circular arc.)

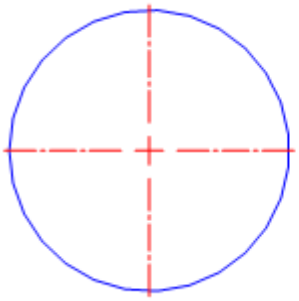
Depending on the current value of the CENTERMARKEXE system variable extension lines are created or not.

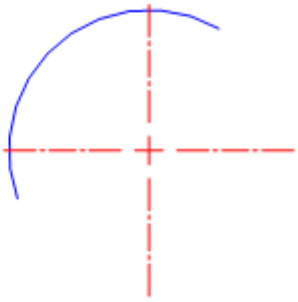




Center mark only

(CENTERMARKEXE = OFF)





Center mark and extension lines Center mark only

(CENTERMARKEXE = ON)

Command Options

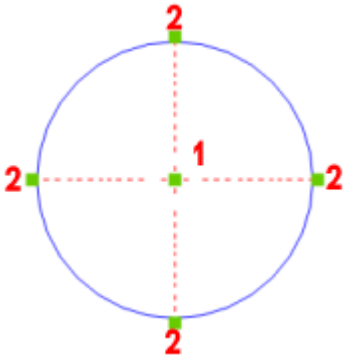
No command options.

Procedures

Defining system variables for center entities

Grips Editing

Center marks can be edited through grips.



1. Center grip: drag the grip to move the center mark*.
2. Overshoot grips: drag the grip to lengthen the extension line**.

*Use the CenterReset command to reset the centerline.
**A lengthened extension line gets a fixed length which is no longer relative to the diameter: changing the diameter does not affect such extension line. It will still be moved along with the circle though.

Related Commands

CenterDisassociate - breaks the association of centerline and a center mark entities.
CenterLine - creates a centerline entity associated with two selected lines.
CenterReassociate - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc
CenterReset - resets centerline and center mark entities.
DimCenter - places center marks at the center points of circles and arcs.

CenterReassociate

Recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

Accessing the Command

command bar: *centerreassociate*

: ***centerreassociate***
Prompts you in the command bar:

Select a center mark or centerline to reassociate: **(Select a center mark or centerline entity.)**

When a center mark is selected:

Select circle or arc: **(Select a circle or a circular arc.)**

or

When a centerline is selected:

Select first line: **(Select a line entity.)**
Select second line: *(Select a line entity.)*

Command Options

No command options.

Related Commands

CenterDisassociate - breaks the association of centerline and a center mark entities.
CenterLine - creates a centerline entity associated with two selected lines.
CenterMark - creates a center mark entity associated with a selected circle or arc.
CenterReset - resets centerline and center mark entities.

CenterReset

Resets centerline and center mark entities.

NOTE

- Centerlines are reset automatically when one of the associated lines is moved or grip edited.
- Center marks are reset automatically when the associated circle or arc is moved or when the radius or diameter has been modified.

Accessing the Command

command bar: *centerreset*

: ***centerreset***
Prompts you in the command bar:

Select a center mark or centerline to reset: **(Select a center mark or centerline entity.)**
Entities in set: 1

Select a center mark or centerline to reset: **(Select a center mark or centerline entity, or press Enter.)**

Command Options

No command options.

Related Commands

CenterDisassociate - breaks the association of centerline and a center mark entities.
CenterLine - creates a centerline entity associated with two selected lines.
CenterMark - creates a center mark entity associated with a selected circle or arc.
CenterReassociate - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc

Chamfer

Creates chamfers; adds an angle to intersections.

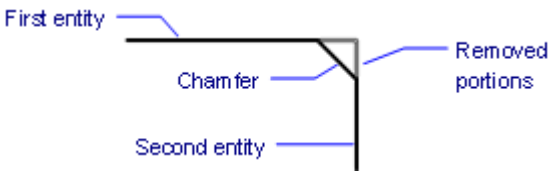
Accessing the Command

command bar: *chamfer*
alias: *cha*
menu bar: *Modify | Chamfer*
toolbar: *Modify |*

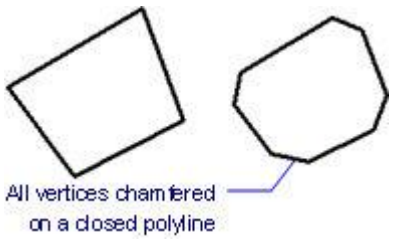
: ***chamfer***
Prompts you in the command bar:

Chamfer (dist1=10, dist2=10) Select first entity [Chamfer Settings...(Settings)/Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple]: **(Select an entity, or enter an option.)**

Select second entity (select with pressed SHIFT to make corner): **(Select another entity.)**



Chamfered intersection.



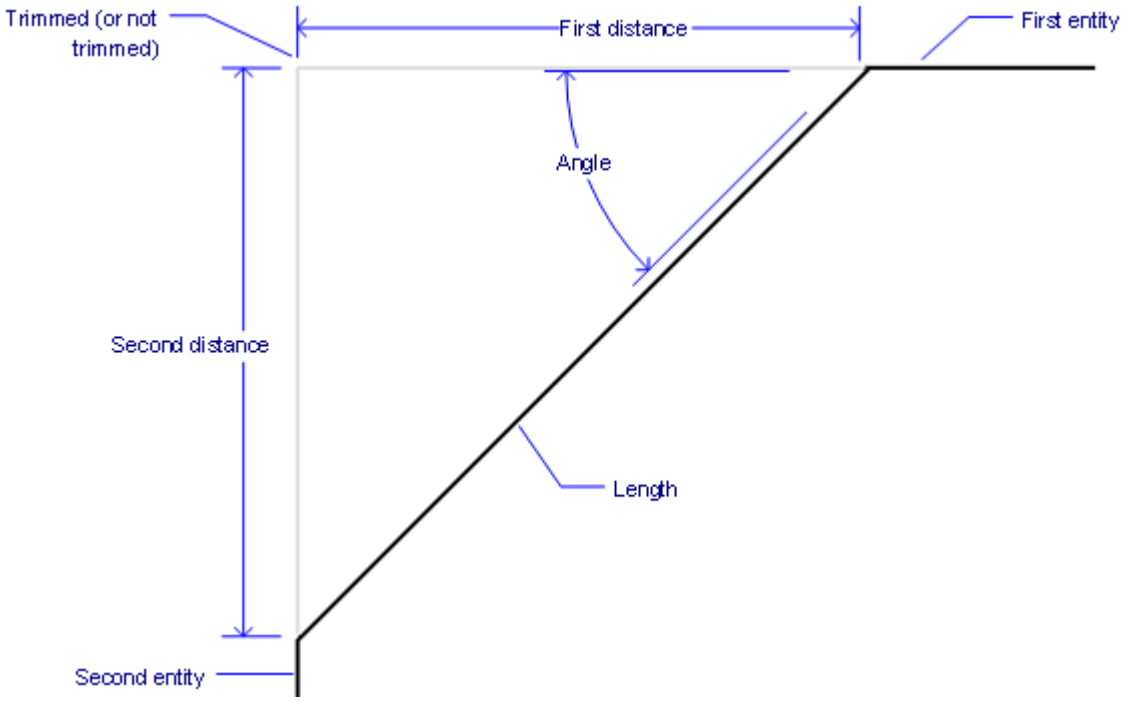
Chamfered polyline.

Command Options

Option	Description
Select first entity	Selects the first entity to be chamfered. Only open entities can be chamfered, with one exception: vertices of closed polylines can be chamfered with the Polyline option. This command cannot place a chamfer between two polylines,but it can place a chamfer between a line and a polyline, even a closed polyline.
Select second entity	Selects the second entity. The two entities need not meet, but cannot be parallel to each other. Press and hold the Shift-key to connect both entities, without creating a chamfer.
Settings	Displays the Settings dialog box for changing chamfer parameters.
Polyline	Chamfers the vertices of a polyline; prompts you: Select 2D polyline to chamfer: *(Select a polyline with at least one vertex.)*
Angle	Prompts you for the Chamfer length and Chamfer angle in the command bar and sets the Chamfer mode to Length-Angle.

Distance	Prompts you for the Chamfer first distance and Chamfer second distance in the command bar and sets the Chamfer mode to Distance-Distance.
Method	Prompts you to set the Chamfer mode in the command bar. The options are: Angle or Distance.
Trim	Sets the trim mode for chamfers and fillets. Prompts you: [Trim/No] <Trim> - choose an option or press Enter to keep the current mode. • Trim: Trims or extends the selected entities. • No: Creates the chamfer or fillet, but leaves the selected entities unchanged.
Undo	Undoes the last chamfer when in Multiple mode.
Multiple	Lets you create multiple chamfers using the same settings.

Settings



Setting	Description
Chamfer Mode	Determines how the chamfer is created: • Distance-Distance - specify two distances. • Length-Angle - specify length and angle.
Chamfer First Distance	Specifies the default value of the first chamfer distance.
Chamfer Second Distance	Specifies the default value of the first second distance.

Chamfer Length	Specifies the default length of the chamfer.
Chamfer Angle	Specifies the default angle of the chamfer.
Trim Mode	Toggles whether intersecting entities are trimmed: • Yes - trims or extends entities to the chamfer line. • No - does not trim intersecting entities.

Procedures

Chamfering using the distance-distance method
Chamfering using the length-angle method
Chamfering all vertices of a polyline

Related Commands

Break - cuts parts out of entities.
dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
Extend - extends open entities to a boundary.
Fillet - fillets intersections.
Settings - specifies parameters for chamfers.
SolidEdit - edits faces, edges, and bodies of solid models.
Trim - trims entities where they cross other entities.

Change

Changes the properties of entities, through the command bar.

Accessing the Command

command bar: *change*
alias: *-ch*

: ***change***
Prompts you in the command bar:

Select entities to change: (Choose one or more entities.)
Entities in set: xx

Select entities to change: (Press Enter to end entity selection.)

Change point [Entities/Properties]: (Enter an option, or specify a new endpoint.)

When you enter 'P' at the prompt:

Property to change
[Color/Elevation/LAyer/LType/ltScale/LWeight/Thickness/TRansparency/Material/Annotative]:
(Enter an option.)

Command Options

Option	Description
Select entities to change	Choose one or more entities in the drawing.
Entities	Specifies change options for rays and infinite lines: Point change for Rays and Infinite Lines affects [Angles/Points/None] <No change> - enter an option: • Angles - changes the angle. • Points - changes the location of the points defining the rays and infinite lines. • None - leaves entities unchanged.
Properties	
Color	Changes the color of the entity; prompts you: New color - enter the name or number of a color.. See Color command.
Elevation	Changes the elevation (z coordinate); prompts you: New elevation - enter a positive or negative number. See Elev command.
LAyer	Changes the layer; layer name must exist in the drawing: New layer - enter name of a layer. See Layer command.
LType	Changes the linetype; linetype must be loaded into the drawing: New linetype - enter the name of a linetype. See Linetype command.
ltScale	Changes the linetype scale; prompts you: New linetype scale - enter a number.

	See *LtScale* sytem variable.

LWeight	Changes the lineweight; prompts you: Enter new lineweight - enter a lineweight unit. See LWeight command.
Thickness	Changes the thickness; prompts you: New thickness - enter a positive or negative number. See Elev command.
Transparency	Changes the transparency; prompts you: New transparency <ByLayer> - enter a value between 0 and 90
Material	Applies a material; prompts you: New material <ByLayer> - enter the name of a material.
Annotative	Sets the Annotative property; prompts you: Make Annotative?[Yes/No] <No> - enter Yes or No.
Change Point	Changes the size of the entity (has no effect on polylines): • Lines - moves an endpoint. • Arcs - changes the length. • Circles - changes the diameter. • Text - changes the position. • Blocks - changes the position

Procedures

To change text entities
To relocate blocks

Related Commands

ChProp - changes the properties of entities in the command bar.
LayCur - moves the selected entities to the current layer.
Properties - changes the properties and sizes of entities in a palette.

ChProp

Changes the properties of entities, with fewer options than the Change command (short for "change properties.")

Accessing the Command

command bar: *chprop*

: ***chprop***
Prompts you in the command bar:

Select entities to change: (Choose one or more entities.)

Select entities to change: (Press Enter to end entity selection.)

Property to change [Color/LAyer/LType/ltScale/LWeight/Thickness/TRansparency/Material/Annotative]: (Enter an option.)

Command Options

Option	Description
Select entities to change	Selects one or more entities in the drawing.
Color	Changes the color of the entity; prompts you: New color - enter the name or number of a color.
Elevation	Changes the elevation (z coordinate); prompts you: New elevation: - enter a positive or negative number.
LAyer	Changes the layer; layer name must exist in the drawing; prompts you: New layer - enter name of a layer.
LType	Changes the linetype; linetype must be loaded into the drawing; prompts you: New linetype - enter the name of a linetype.
ltScale	Changes the linetype scale; prompts you: New linetype scale - enter a number.
LWeight	Changes the lineweight; prompts you: Enter new lineweight - enter a lineweight unit.
Thickness	Changes the thickness; prompts you: New thickness - enter a positive or negative number.
Transparencey	Changes the transparency; prompts you: New transparency <ByLayer> - enter a value between 0 and 90
Material	Applies a material; prompts you: New material <ByLayer> - enter the name of a material.
Annotative	Sets the Annotative property; prompts you: Make Annotative?[Yes/No] <No> - enter Yes or No.

Related Commands

Change - changes the properties and sizes of entities in the command bar.
LayCur - moves the selected entities to the current layer.
Properties - changes the properties and sizes of entities in a palette.

ChSpace

Moves entities from paper space to model space and vice versa.

Accessing the Command

The command applies to layouts only.
When executed in model space, a message displays in the command bar: This command can only be used in a Layout Tab ('Paper Space').

Moving entities from paper space to model space

Select the layout that contains the entities to be moved.

command bar: *chspace*
menu: *Modify | Change Space*

: ***chspace***

Prompts you in the command bar:

Select entities: (Select an entity.)

Entities in set: 1

Select entities: (Select more entities or press Enter.)

Select TARGET viewport and press ENTER to continue: (Click inside a viewport or press Enter to accept the currently selected viewport.)

<number of objects> objects were moved from PAPER space to MODEL space. Objects were scaled to maintain visual appearance by a factor of <scale factor>.

The scale factor is the inverse of custom scale of the selected target viewport. E.g. if the custom scale = 1/2 (0.5), scale factor = 2.

Moving entities from model space to paper space

When in a layout, double click a viewport where the entities, which need to be moved, display.

command bar: *chspace*

: ***chspace***

Prompts you in the command bar:

Select entities: (Select an entity.)

Entities in set: 1

Select entities: (Select more entities or press Enter.)

Select SOURCE viewport and press ENTER to continue: (Click inside a viewport or press Enter to accept the currently selected viewport.)

<number of objects> objects were moved from MODEL space to PAPER space. Objects were scaled to maintain visual appearance by a factor of <scale factor>

The scale factor equals the custom scale of the selected source viewport.

Command Options

No options.

Related Commands

CopyClip - copies entities to the Clipboard.
CutClip - copies entities to the Clipboard, and then erases them from the drawing.

MSpace - switches to model space.
PasteClip - pastes entities from the Clipboard into the drawing (short for "paste clipboard").
PSpace - switches to paper space.

Circle

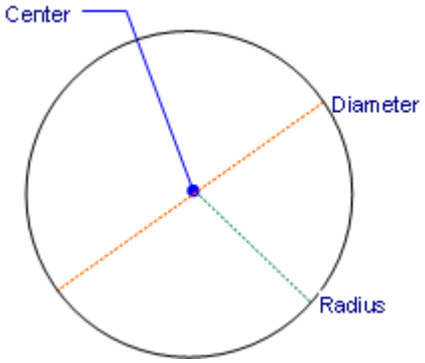
Draws circles by several methods.

Accessing the Command

command bar: *circle*
alias: *c*
menu bar: *Draw | Circle*
toolbar: *Draw |*

: ***circle***

Prompts you in the command bar:
Select center of circle or [**2Point**/**3Point**/**TTR**/**Arc**/**Multiple**]: *(Pick a point, or enter an option.)*
Set Radius or [Diameter] <x.xx>: *(Pick a point, enter a distance, or type D to define the diameter.)*

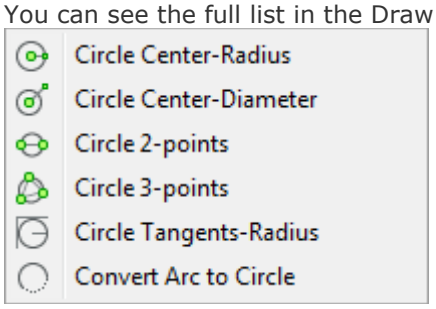


Circle by center and diameter or radius.

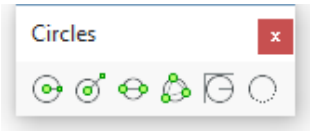
This command also draws circles by two points, three points, and by tangent, tangent, and radius.

It also converts arcs to circles.

You can see the full list in the Draw | Circle menu:

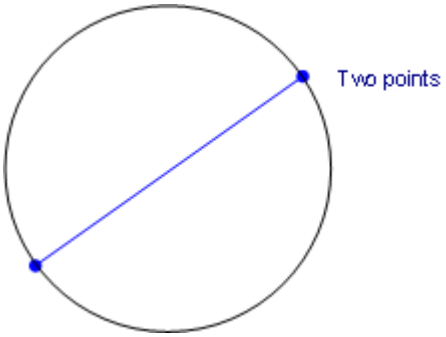
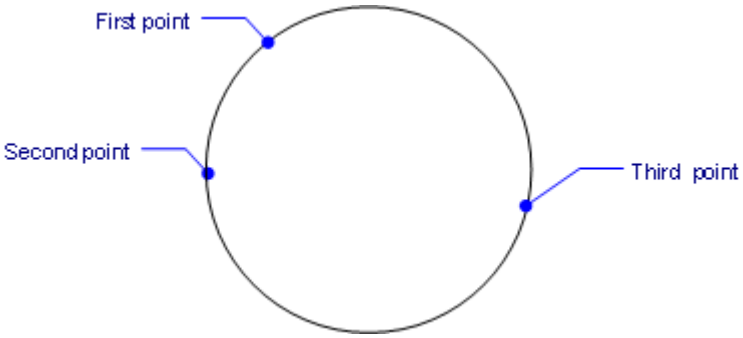


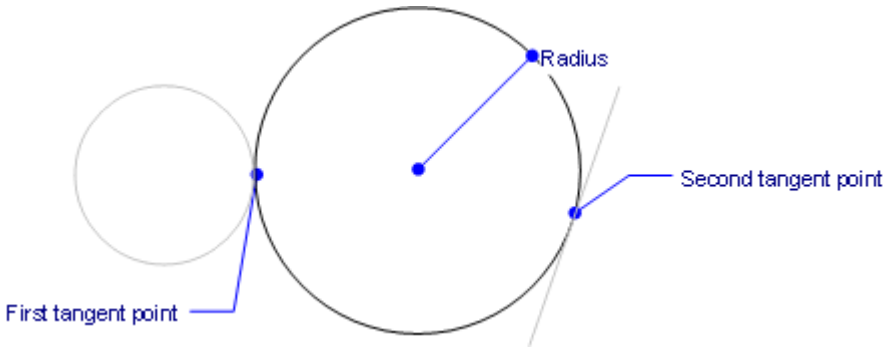
and the Circles toolbar:



Command Options

Option	Description
Center of circle	Specifies the center of the circle; enter x,y,z

	coordinates, or pick a point in the drawing.
Diameter	Specifies the diameter of the circle; prompts you: Diameter of circle - enter a distance, coordinates, or pick a point.
Radius	Specifies the radius of the circle; prompts you: Radius of circle - enter a distance, coordinates, or pick a point.
2Point  	Defines the circle by two points on its circumference; prompts you: First point on diameter - enter coordinates, or pick a point. Second point on diameter - enter coordinates, or pick another point. The abbreviation for this option is 2P.
3Point  	Defines the circle by three points on its circumference; prompts you: First point on circle - enter coordinates, or pick a point. Second point - enter coordinates, or pick another point. Third point - enter coordinates, or pick another point. The abbreviation for this option is 3P.

RadTanTan  	Defines the circle by two points tangent to other entities and the radius; prompts you: Select first tangent point - enter coordinates, or pick an entity. Select second tangent point - enter coordinates, or pick another entity. Radius of circle - specify distance, or pick two points. The abbreviation for this option is RTT. The *Tangent* entity snap is activated automatically when you are prompted to identify the tangent points.
Arc	Converts arcs to circles by closing them; prompts you: Select arc to convert to circle: (Choose an arc.) See **Arc** command.
Multiple	Repeats the command until you press Esc.

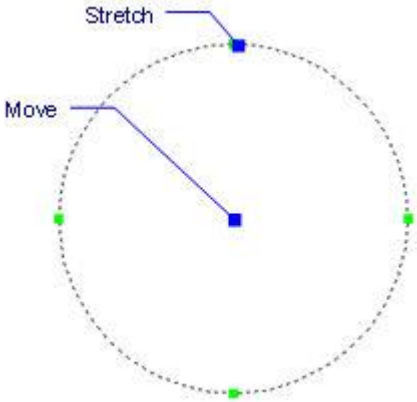
Procedures

General procedure to draw a circle
Editing a circle

Grips Editing

Circles can be edited directly through grips:

1. Select the circle. Notice that it has five grips.
2. Drag the grips to perform the following actions:
    - Drag the quadrant grips to stretch the circle (increase and decrease the radius).
    - Drag the center point grip to move the circle.



Related Commands

Ai_CircTan - draws a circle tangent to three entities.
Arc - draws arcs.
Change - changes the diameter of circles.
Ellipse - draws ellipses.
Join - joins arcs to create circles.
Donut - draws fat circles from polyline arcs.
DimRadius and DimDiameter - dimensions circles.

Center - snaps to the center of circles.
Quadrant - snaps to the 90-degree points on the circumference of circles.
Tangent - snaps to the tangency of circles.

CleanScreenOff

Displays user interface elements that were hidden by the CleanScreenOn command.

Accessing the Command

command bar: *cleanscreenoff*
Quad: *General | Clean Screen*
keyboard: *Ctrl + 0* (zero)*

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

: ***cleanscreenoff***

Displays no prompts in the command bar.

Related Commands

CleanScreenOn - hides elements of the user interface.
CommandLine - displays the command bar.
CommandLineHide - hides the command bar.
Ribbon - displays the ribbon.
RibbonClose - hides the ribbon.
StatBar - toggles the display of the status bar.

CleanScreenOn

Hides elements of the user interface.

Accessing the Command

command bar: *cleanscreenon*
menu bar:
toolbar: *Standard* |
Quad: *General* | *Clean Screen*
keyboard: *Ctrl + 0* (zero)*

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

: cleanscreenon
Displays no prompts in the command bar:

Elements in the user interface are hidden according to the current value of the CLEANSCREENOPTIONS system variable:

Clean screen options	0x007F (127)
1	☑ Hide document tabs
2	☑ Hide dockable panels
4	☑ Hide toolbars
8	☑ Hide ribbon
16	☑ Hide command line panel
32	☑ Hide status bar
64	☑ Hide menu bar

Command Options

Option	Description
Hide document tabs	The display of document tabs is controlled by the ShowDocTabs user preference. See Working with document tabs in the User Guide.
Hide dockable panels	The following panels can be docked: Command Bar, BIM Compositions, Content Browser, Layers, Mechanical Browser, Properties Bar, Render Materials, Ribbon, Sheet Sets, Structure, Tool Palettes See the Dockable Panels topic in the User Guide.
Hide toolbars	Hides all toolbars. See the Toolbars topic in the User Guide.
Hide ribbon	The display of the ribbon is managed by the RIBBON and RIBBONCLOSE commands.
Hide command line panel	The display of the commandline panel is managed by the COMMANDLINE and COMMANDLINEHIDE commands.

Hide status bar	The STATBAR command toggles the display of the status bar ON/OFF. The current status of the status bar is saved through the WNDLSTAT system variable.
Hide menu bar	The display of the menu bar is controlled by the MENUBAR system variable.

Related Commands

CleanScreenOff - displays user interface elements that were hidden by the CleanScreenOn command.
CommandLine - displays the command bar.
CommandLineHide - hides the command bar.
Ribbon - displays the ribbon.
RibbonClose - hides the ribbon.
StatBar - toggles the display of the status bar.

CleanUnusedVariables

Purges variables not used in constraint expressions and not linked to dimensions.

Accessing the Command

command bar: *cleanunusedvariables*
toolbar: *Parameters* |

: ***cleanunusedvariables***

When no unused variables exist in the drawing a message displays:
** No unused variables detected

Command Options

No options.

Related Commands

-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

Delconstraint - removes all dimensional and geometrical constraints from an entity.

ClipDisplay

Toggles the Clip Display property of a section plane or a BIM section entity.

Accessing the Command

command bar: *clipdisplay*
mouse: double click a BIM section entity.

: ***clipdisplay***
Prompts you in the command bar:
Select a section entity: *(Select a section plane or a BIM section entity.)*

Command Options

No command options.

Procedures

Using Clip Display

Related Commands

BimSection - creates a BIM section entity.
LiveSection - toggles the Live Section property of a section plane.
SectionPlane - creates a section entity that creates sections of 3D solids.

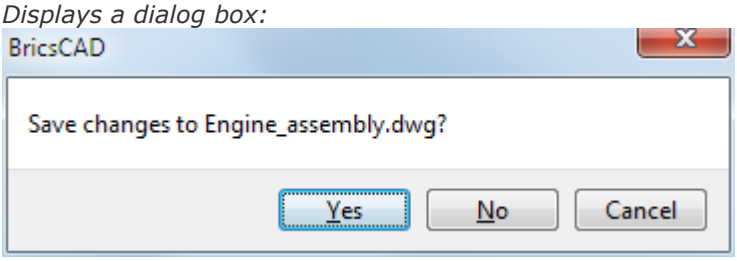
Close

Closes the current drawing.

Accessing the Command

command bar: *close*
shortcut: *Ctrl+F4*
menu bar: *File | Close*

: ***close***
Displays a dialog box:



Click the Yes, No, or Cancel button.

Command Options

Option	Description
Yes	Saves the drawing before closing it.
No	Does not save the drawing, and then closes it.
Cancel	Cancels the command and returns to the drawing.

Procedures

Working with document tabs

Related Commands

Open - opens drawing files.
WClose - closes the current window.
Quit - exits BricsCAD after closing open drawings.

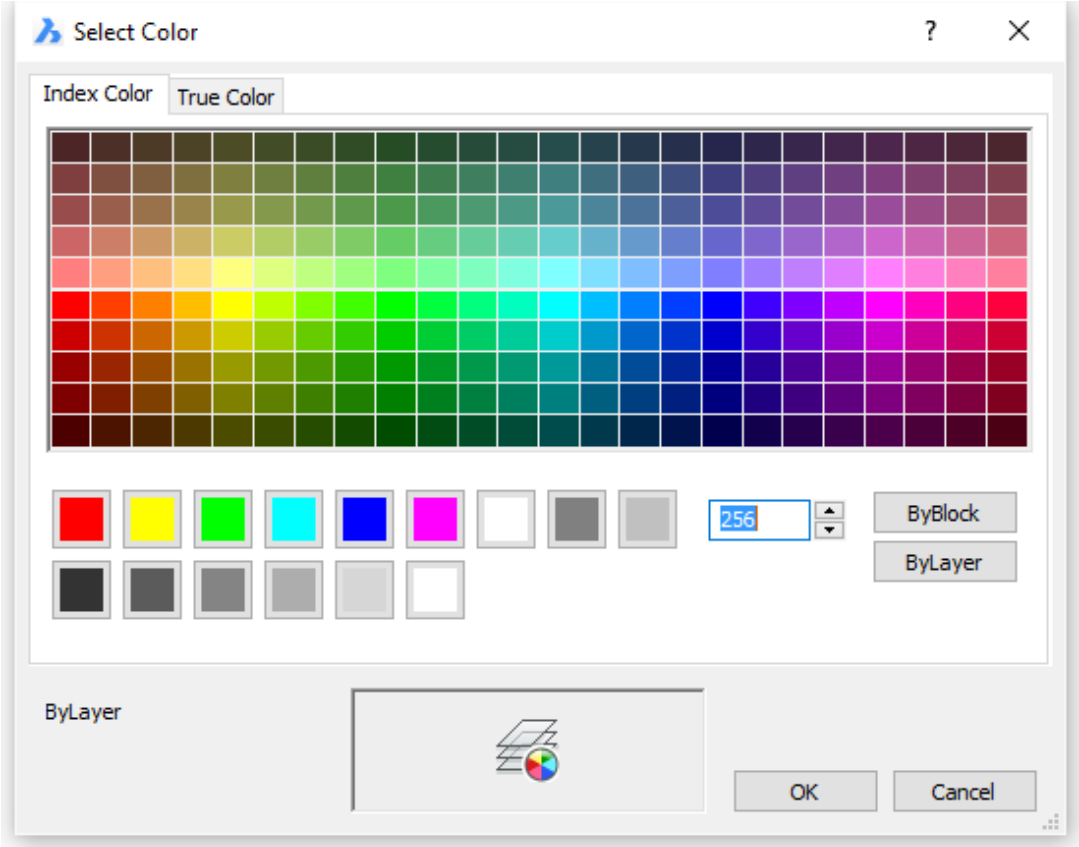
Color

Specifies the current color, through a dialog box.

Accessing the Command

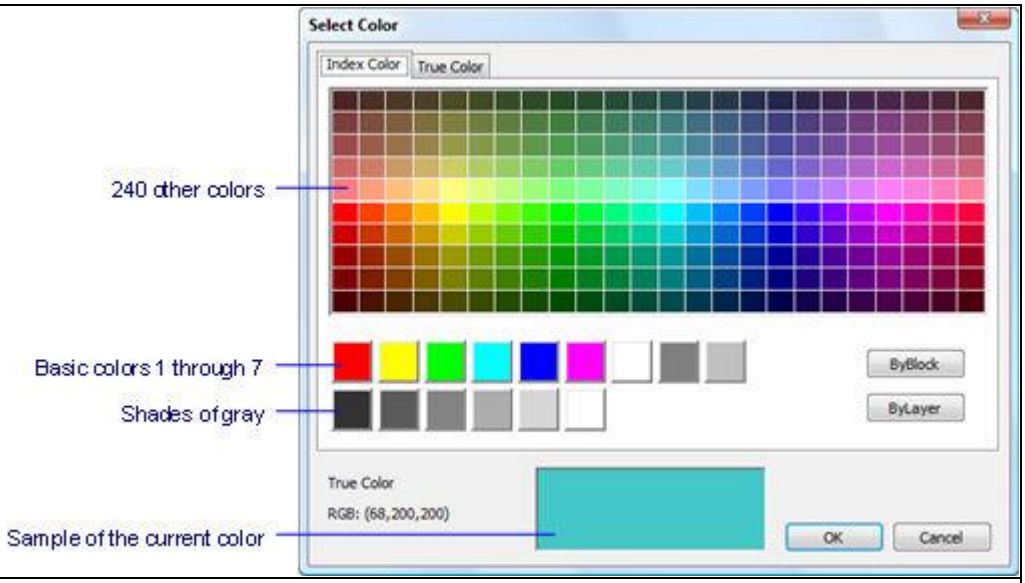
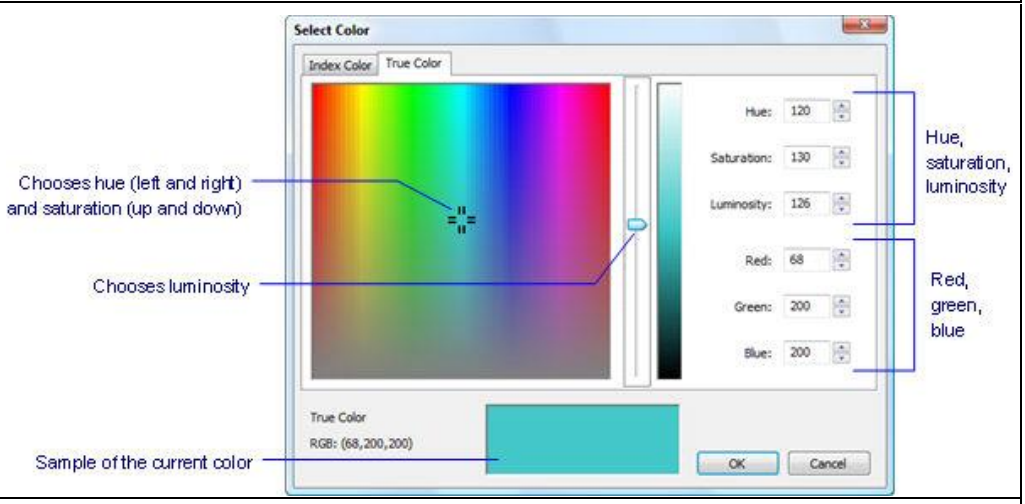
command bar: *color*
transparent: *'color*
aliases: *col, color, ddcolor, ddcolour, setcolor*
menu bar: *Settings | Color | Select Color*
toolbar: *Settings |*
status bar: *Color* field (*ByLayer or*)

: ***color***
Displays a dialog box:



Choose a color, and then click **OK**.

Command Options

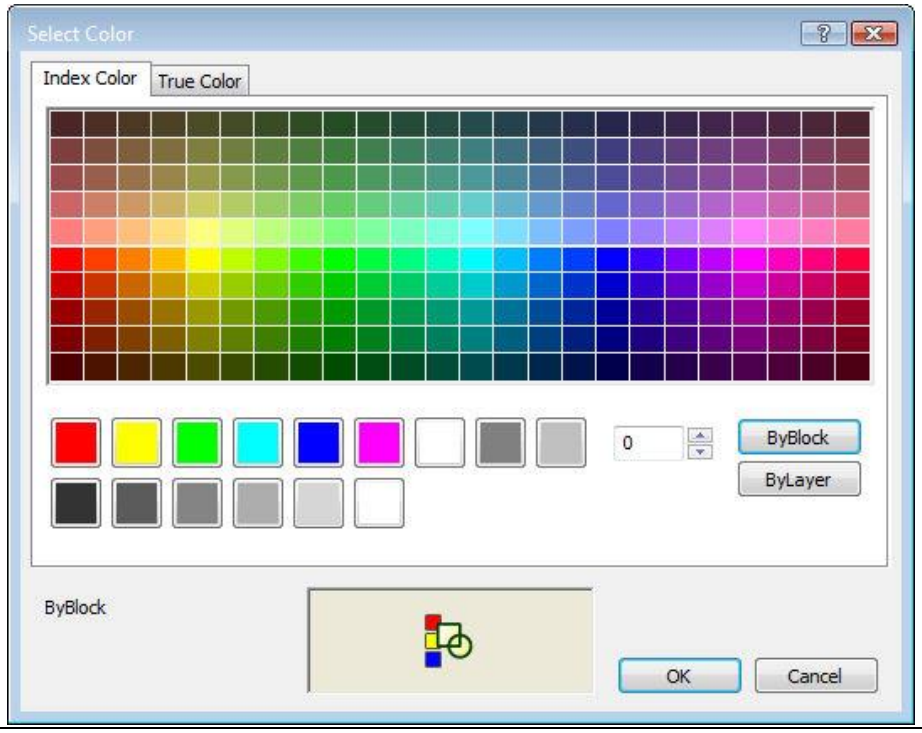
Index Color tab	
ByBlock	Sets the color to ByBlock. Entities take on the color defined by their block.
ByLayer	Sets the color the ByLayer. Entities take on the color defined by their layer's property.
True Color tab	
Hue	Specifies the color's hue; range is from 0 to 240. **Hue** is the color, ranging from red through green, blue, and pink.
Saturation	Specifies the intensity of the color; range is from 0 to 240. **Saturation** is the intensity of the color, where 0 = gray and 240 = full color.
Luminosity	Specifies the brightness of the color; range is from 0 to 240. **Luminosity** is the lightness or darkness, where 0 = black and 240 = white. Hue, saturation, and luminosity work together, while red, green, and blue work separately.
Red	Specifies the amount of red in the color; range is from 0 (no red) to 255 (full red).

Green	Specifies the amount of green in the color; range is from 0 (no green) to 255 (full green).
Blue	Specifies the amount of blue in the color; range is from 0 (no blue) to 255 (full blue).

Status Bar Menu

Right-click ByLayer (or other color name) on the status bar for the shortcut menu:

Shortcut Menu	Description
ByLayer, ByBlock, Red, et cetera	Chooses the default color.
Properties	Opens the Select Color dialog box.


Select Color
Index Color
True Color
0
ByBlock
ByLayer
ByBlock
OK
Cancel

Related Commands

-Color - sets the working color through the command bar.
DimStyle and -DimStyle - specify colors for dimensions.
Layer and -Layer - specify colors for layers.
PlotStyle - specifies how colors control plotted output.
Settings - specifies color-related parameters, such as the colors of grips, background, UCS and cursor axes, snap markers, and so on.

-Color

Sets the current color, through the command bar.

Accessing the Command

command bar: *-Color*
aliases: *-col, -colour*
status bar: *ByLayer*

: ***-color***
Prompts you in the command bar:
Color for drawing new entities <white>: *(Enter a color name, number, or RGB value.)*

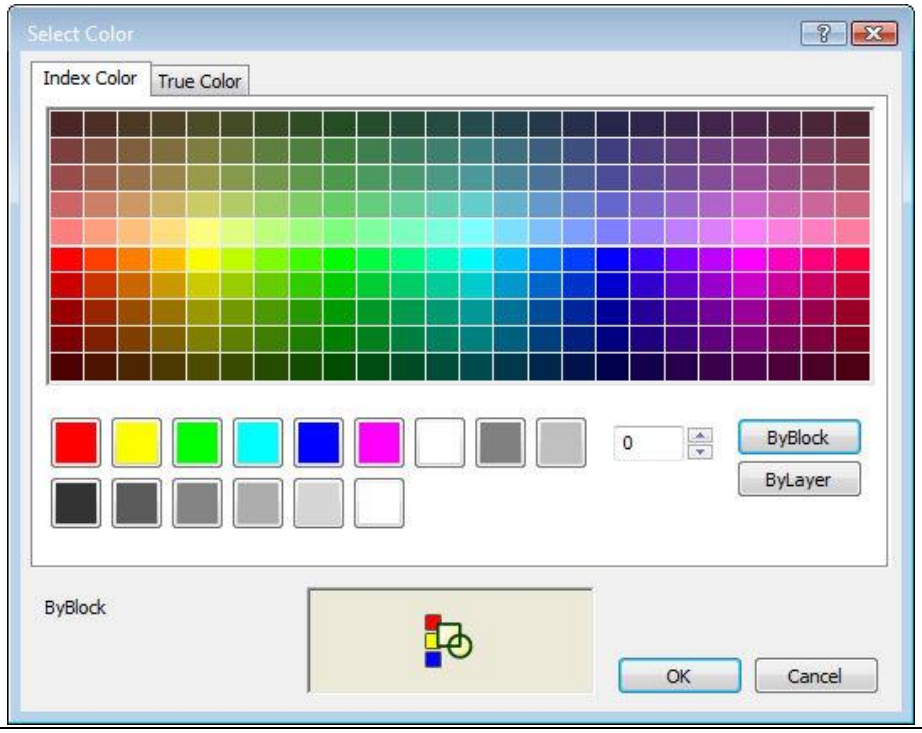
Command Option

Option	Description
Color for new drawing entities	Enter the name, number, or RGB value of a color: • Names - specifies red (color #1) , yellow (2), green (3), cyan (4), blue (5), magenta (6), white (7), ByLayer (256), or ByBlock (0). • Numbers - specifies a number from 0 to 256. • RGB - specifies a value for each of red, green, and blue ranging from 0 to 255. For example, white is 255,255,255 and gray is 128,128,128.

Status Bar Menu

Right-click ByLayer (or other color name) on the status bar for the shortcut menu:

Shortcut Menu	Description
ByLayer, ByBlock, Red, et cetera	Chooses the default color.
Properties	Opens the Select Color dialog box. See Color command.



Related Commands

Color - sets the working color through a dialog box.
DimStyle and -DimStyle - specify colors for dimensions.
Layer and -Layer - specify colors for layers.
PlotStyle - specifies how colors control plotted output.
Settings - specifies color-related parameters, such as the colors of grips, background, UCS and cursor axes, snap markers, and so on.

CommandLine

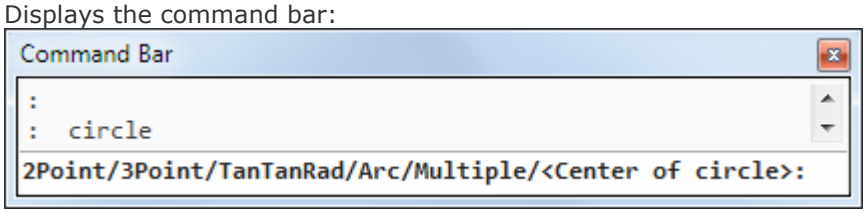
Displays the command bar.

Accessing the Command

command bar: *commandline*
transparent: *'commandline*
menu bar: *View | Command Bar*
mouse: double clicking the left side of the Status Bar toggles the display of the Command Bar

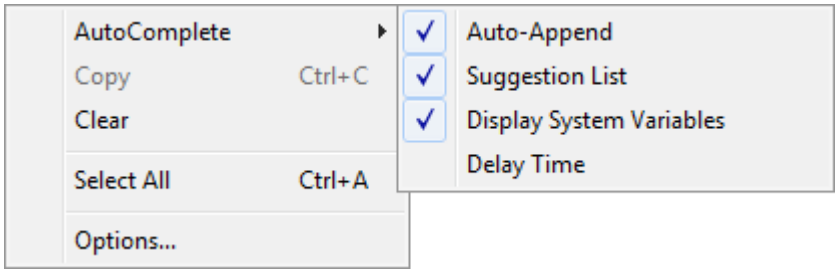
: ***commandline***

Displays the command bar:



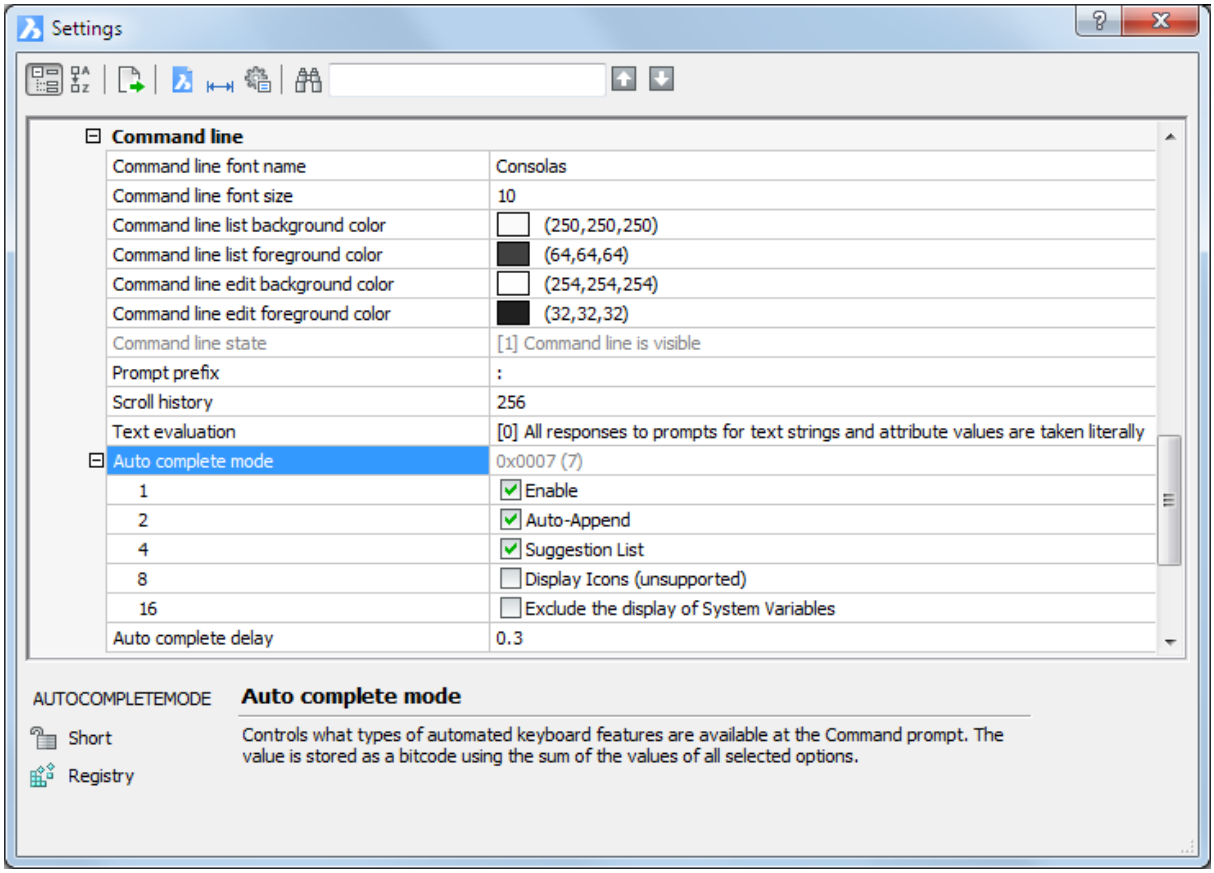
Shortcut Menu Options

Right-click to display the following context menu:



Option	Description
AutoComplete	**Auto-Append:** If checked, automatically completes the entry if only one possible command is left. **Suggestion List:** If checked, displays a list of possible commands. **Display System Variables:** If checked, includes system variables in the suggestion list. **Delay Time:** Sets the time in seconds before the automated keyboard features display.
Copy	Copies the selected text to the Clipboard.
Clear	Clears the complete prompt history
Select All	Selects all the complete content of the prompt history
Options	Displays the *Settings* dialog box: Command line.

Command line options dialog:



Related Commands

CleanScreenOff - displays user interface elements that were hidden by te CleanScreenOn command.

CleanScreenOn - hides elements of the user interface.

CommandLineHide - hides the command line.

GraphScr - brings the graphics window to the foreground.

Scrollbar - toggles the display of the scroll bars.

StatBar - toggles the display of the status bar.

TextScr - brings the text window to the foreground.

Toolbar and -Toolbar - toggle the display of toolbars.

CommandLineHide

Hides the command bar.

Accessing the Command

command bar: *commandlinehide*
transparent: *'commandlinehide*
menu bar: *View | Command Bar*
mouse: double clicking the left side of the Status Bar toggles the display of the Command Bar.

: **commandlinehide**
Hides the command bar. Prompts are displayed on the status bar.

: commandline	-0.1477,3.4426,0.0000	0	ByBl

Command Options

No options.

Related Commands

CleanScreenOff - displays user interface elements that were hidden by te CleanScreenOn command.

CleanScreenOn - hides elements of the user interface.

CommandLine - shows the command line.

GraphScr - brings the graphics window to the foreground.

Scrollbar - toggles the display of the scroll bars.

StatBar - toggles the display of the status bar.

TextScr - brings the text window to the foreground.

Toolbar and -Toolbar - toggle the display of toolbars.

Commands

Lists the names of commands. For each command the original command name is followed by the translated command name, which is the same in the English version.

Accessing the Command

command bar: *commands*

: **commands**
Prompts you in the command bar:
List commands [Internal/External/Both] <Internal>: *(Press Enter to list internal commands only, or enter an option.)*

Command Options

Option	Description
Internal	Lists command names native to BricsCAD.
External	Lists command names from add-on applications.
Both	List both internal and external command names.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

CommunicatorInfo

Checks whether the Communicator product is correctly installed and prints a diagnostic report in the Prompt History window.
The command is available on Windows only. The Communicator version must match the major version of BricsCAD

Accessing the Command

command bar: *communicatorinfo*

: ***communicatorinfo***
Prints a diagnostic report in the Prompt History window:

```
: COMMUNICATORINFO

MSVC runtime version 10: OK.
MSVC runtime version 11: OK.
MSVC runtime version 12: OK.
BricsCAD path: 'C:\Program Files\Bricsys\BricsCAD V17 en_US\'.
ACIS: OK.
ACIS: dll product version '20171.0.1.17023'.
Communicator installation path: 'C:\Program Files\Bricsys\BricsCAD Communicator V17\'.
Communicator InterOp check: OK.
Communicator Connect license check: OK.
Communicator UConnect license check: OK.
Using PS Direct translation mode.
```

Command Options

No options.

Related Commands

Import - imports the geometry from external files into the current drawing.
Export - exports drawings in a variety of vector and raster formats.

Cone

Creates a cone as polygon mesh (Classic); creates a cone as a 3D solid (Pro and Platinum).

Accessing the Command

command bar: *cone*
menu bar: *Model | 3D Solids | Cone*
toolbar: *3D Solids |*

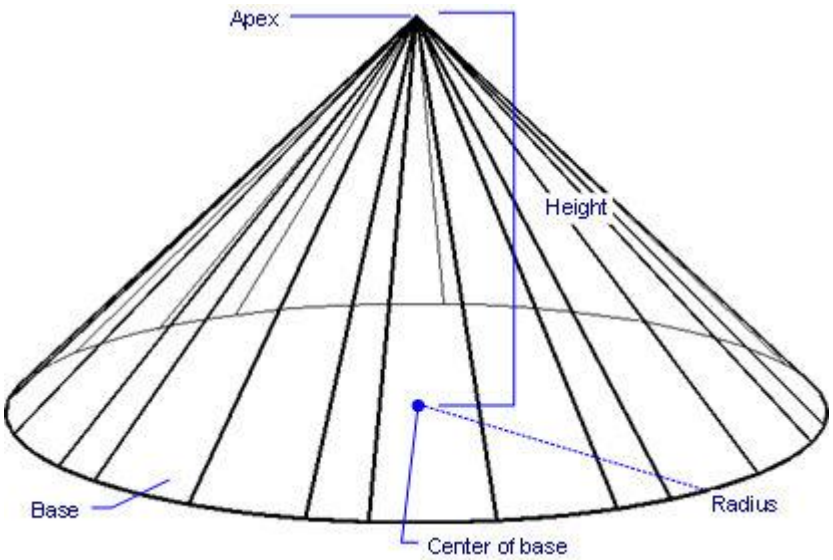
: ***cone***

Prompts you in the command bar:

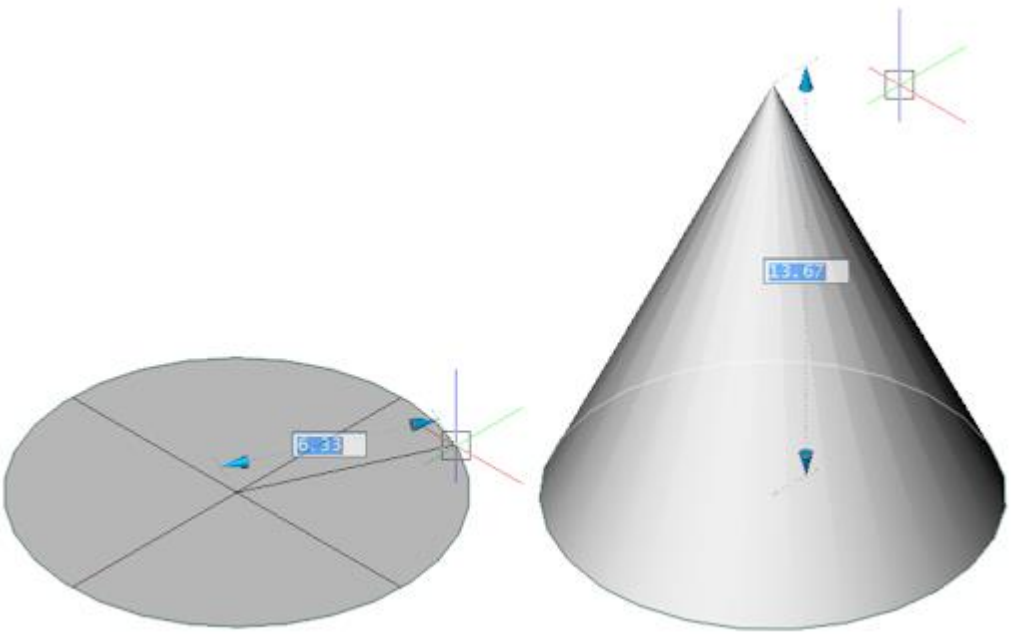
Select center point or [**3Point**/**2Point**/**Ttr**/**Elliptical**]: *(Pick a point or enter an option.)*

Radius of base of cone or Diameter <xx.xx>: (Press Enter to accept the default value, type a value or pick a point to define the base radius, or type D to define the diameter.)

Specify height or [**2Point**/**Axis**/**Top**] <xx.xx>: (Press Enter to accept the default value, type a value or pick a point, or enter an option.)



If Dynamic Dimensions are active, you can enter the dimensions in the dynamic dimensions entry fields.



Command Options

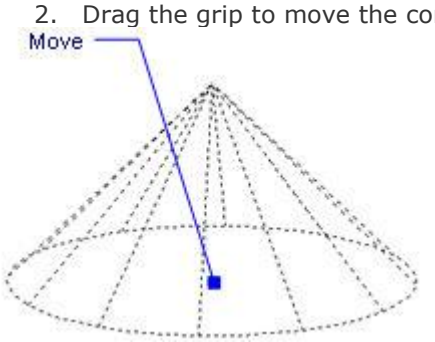
Option	Description
Base options	
3Point	Defines a circular base through 3 points. Prompts you: First point: Specify a point. Second point: Specify a point. Third point: Specify a point. Specify height or [2Point/Axis/Top] <xx.xx>: **Type a value or choose an option.**
2Point	Defines a circular base through 2 points (= diameter). Prompts you: First end of diameter: Specify a point. Second end of diameter: Specify a point. Specify height or [**2Point**/**Axis**/**Top**] <xx.xx>: Type a value or choose an option.
TTR (Tangent Tangent Radius)	Defines a circular base tangent to two entities and a radius. Prompts you: Specify point on object for first tangent: Select a point on a circle, arc, line or polyline segment*. Specify point on object for second tangent: Select a point on a circle, arc, line or polyline segment*. Radius of circle: Type a value or specify two points. Specify height or [**2Point**/**Axis**/**Top**] <xx.xx>: Type a value or choose an option. *Use theTangent entity snap..
Elliptical	Defines an elliptical base. Prompts you: Set first end of ellipse or [Center]: Specify the first end of the major axis of the ellipse.

	Second end of ellipse axis: Specify the second end of the major axis of the ellipse. Other end of ellipse axis: Specify the endpoint of the minor axis of the ellipse. Specify height or [2Point/Axis/Top] <xx.xx>: **Type a value or choose an option.**
Height options	
2Point	Defines the height of the cone as the distance between two points.
Axis	Allows to define the axis of the cone. The height of the cone is the distance between the center point of the base and the identified point. If necessary the base of the cone is rotated . Prompts you: Specify axis endpoint: Specify a point.
Top	Allows to create a truncated cone. Prompts you: Radius of top of cone <0>: Specify the radius of the top face of the truncated cone. Specify height or [**2Point**/**Axis**/**Top**] <xx.xx>: Press Enter to accept the default value, type a value or pick a point, or enter an option.

Grips Editing

Cone can be edited directly through grips:

1. Select the Cone. Notice that it has one grip.
2. Drag the grip to move the cone.



Related Commands

Box - creates 3D boxes as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.

Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

ConstraintBar

Controls the display of the constraint bar of entities.

Accessing the Command

command bar: *constraintbar*
menu bar: *Parametric | Show/Hide Geometrical Constraint*
toolbar: *2D Constraints |*

: ***constraintbar***
Prompts you in the command bar:

Select entities: **(select entities.)**
Entities in set: 1

Select entities: **(select more entities or press Enter.)**

Select option for constraints [Show/Hide/Reset] <Show>: **(select an option)**

Command Options

Option	Description
Show	The Constraint bar of the selected entities displays*.
Hide	The Constraint bar of the selected entities is hidden*.
Reset	Relocates the constraint bar to its default position (near the midpoint of the entity).

* Constraint Bars are always hidden when opening a drawing.

Procedures

To set the display of constraint bars
To control the position of a constraint bar

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - Removes all dimensional and geometrical constraints from an entity.

ContentBrowserclose

Closes the Content Browser panel.

Accessing the Command

command bar: *contentbrowserclose*

: ***contentbrowserclose***

Command Options

No options.

Related Commands

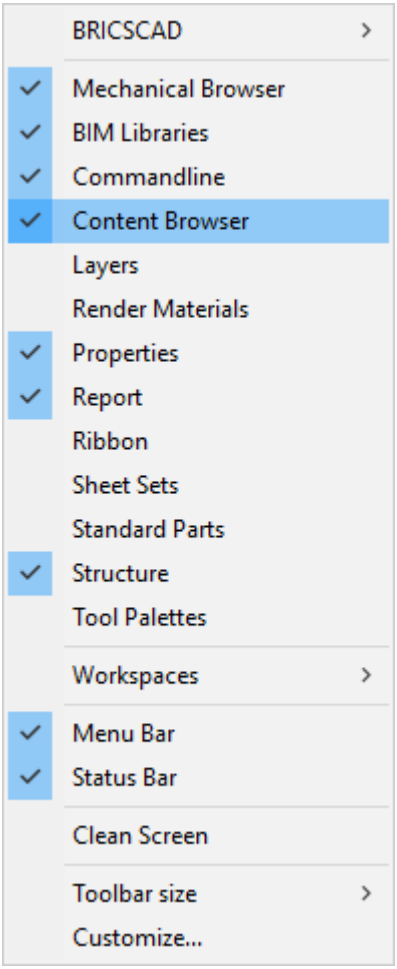
ContentBrowserOpen - opens the Content Browser panel.

ContentBrowserOpen

Opens the Content Browser panel.

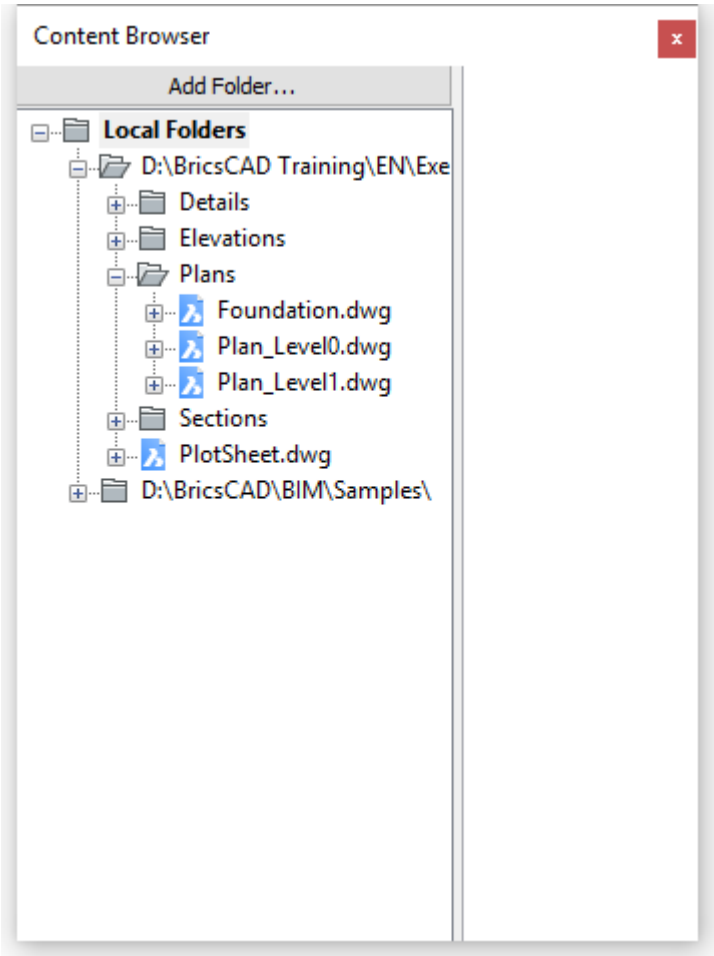
Accessing the Command

command bar: *contentbrowseropen*
context menu*: *Content Browser*



* Place the cursor over a toolbar or a ribbon panel, then right click.

: ***contentbrowseropen***
Displays the Content Browser panel:

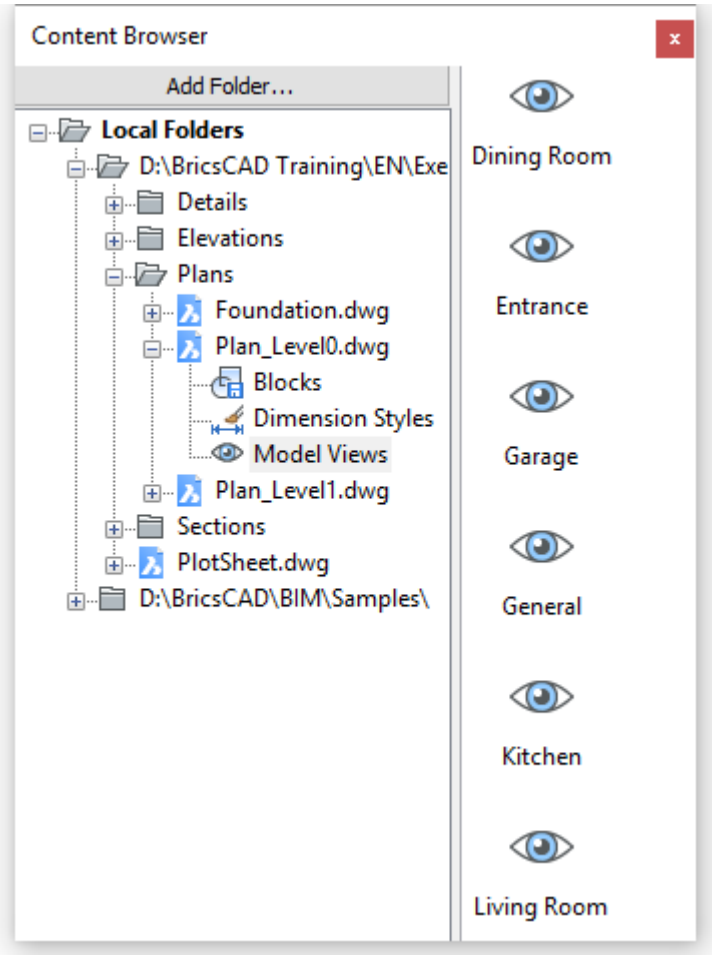


Command Options

Option	Description
Add Folder...	Adds a folder to the *Local Folders* tree. Do one of the following: • Click the *Add Folder...* button. • Right click on the *Local Folders* node, then choose *Add folder* in the context menu.
Remove Folder	Removes a folder from the *Local Folders* tree. Right click the folder you want to remove and choose *Remove folder* in the context menu.
Open Drawing	Opens a drawing in one of the folders in the *Local Folders* tree. Do one of the following: • Double click the drawing name. • Right click the drawing name and choose *Open drawing* in the context menu.
Insert Block	Executes the -Insert command. Do the following: • Expand the source drawing in the *Local Folders* tree. • Click *Blocks*. • Select the block, then do one of the following:

	• Drag the block onto the current drawing. • Right click and choose *Insert Block* in the context menu.

Copy Dimension Style	Copies the selected dimension style(s) to the current drawing.
	Do the following:
	• Expand the source drawing in the *Local Folders* tree.
	• Click *Dimension Styles*.
	• Select the style(s)*, then do one of the following:
	  • Drag the selected style(s) onto the current drawing.
	  • Right click and choose *Add Dimstyle(s)* in the context menu.
	* Press and hold the Ctrl-key to select multiple dimension styles.

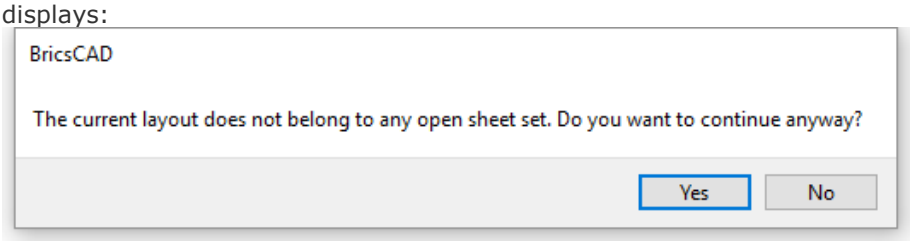
Place View	Executes the PlaceView command. Places a model view selected from a source drawing into a paper space layout of the current drawing. Do the following: • Switch to the layout where you want to place the model view in the target drawing. • Expand the source drawing in the *Local Folders* tree. • Click *Model Views*.  • Drag a model view from the *Content Browser* panel onto the paper space layout of the target drawing.* Prompts you:

Enter file name: D:\BricsCAD Training\Bricscad\TrainingCD\EN\Exercises\House.dwg
Enter view name to place or [?]: Garage
(Option) right click to select scale.
Enter view origin point:

- (option) Right click and choose a scale in the Scale List
- Click a point to place the view.

The source drawing is inserted as an Xref drawing in the model space of the current drawing, on a layer with the same name as the source drawing
A paper space viewport is added looking onto the xref according to a newly added view that matches the source view.
In this new viewport, only the layers of the xref are set visible, so the view is not disturbed by other drawing content.
If the current drawing belongs to a sheetset, a matching sheetset view is created and a view label block is added.

* If the current layout does not belong to an open sheet set an alert box displays:



Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

ContentBrowserclose - closes the Content Browser panel.
-Insert - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.
MView - creates one or more viewports in layout tabs.
PlaceView - places a model view selected from a source drawing into a paper space layout of the current drawing.

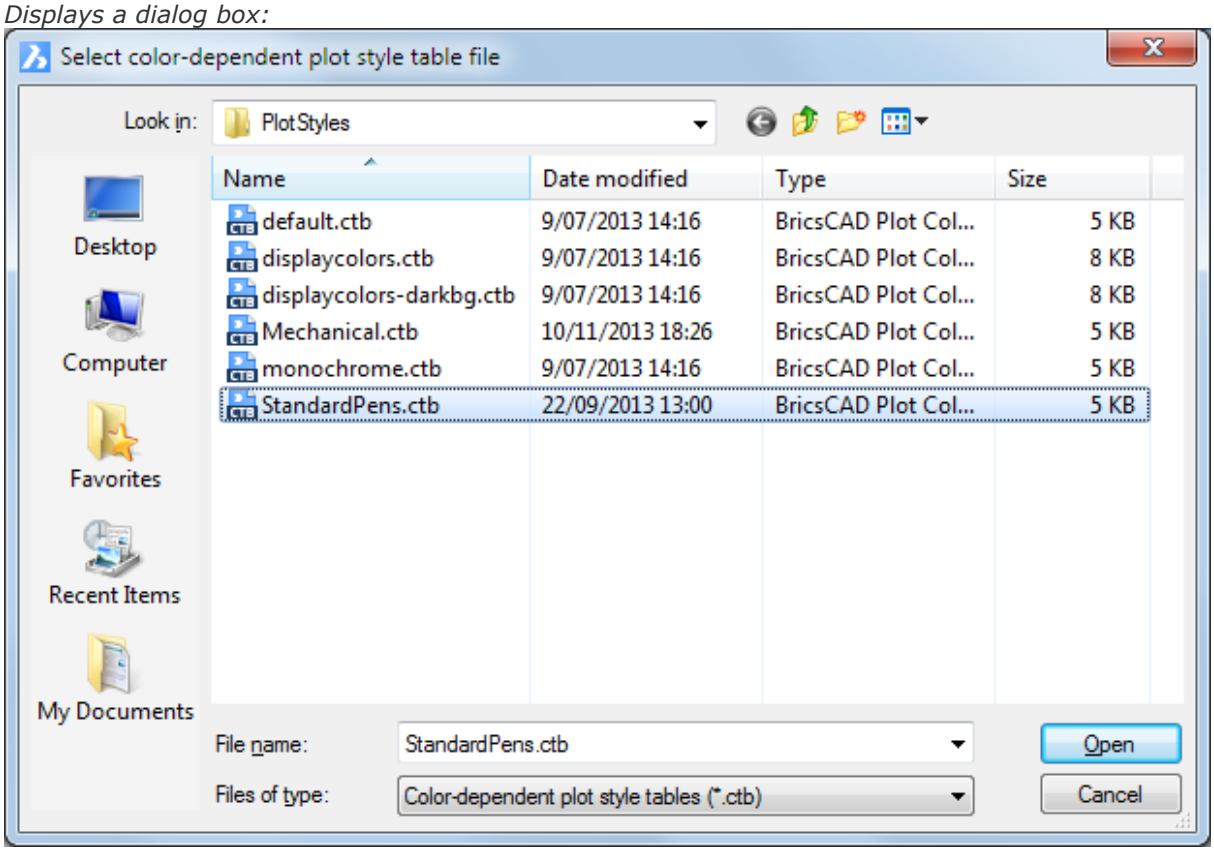
ConvertCtb

Converts older CBT (color-based plot tables) files to newer STB (style-based plot tables) files.

Accessing the Command

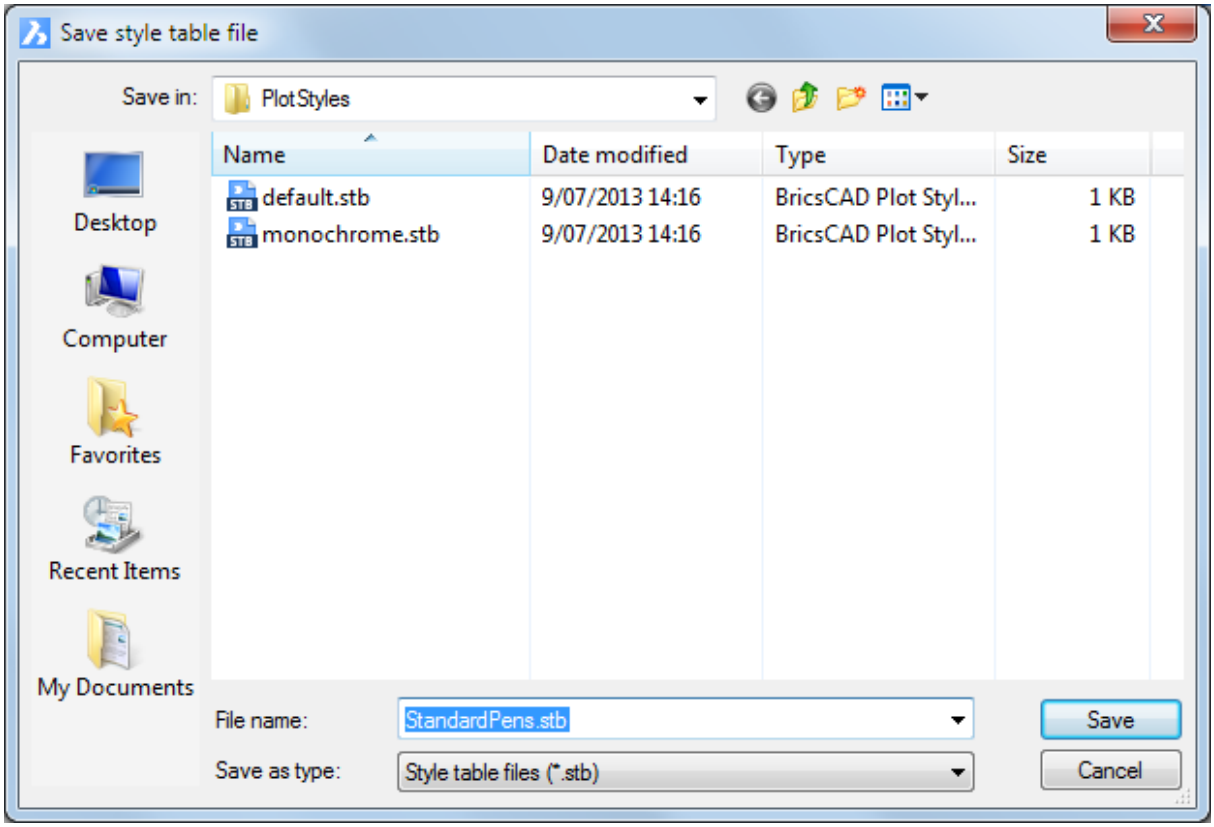
command bar: *convertctb*

: ***convertctb***
Displays a dialog box:



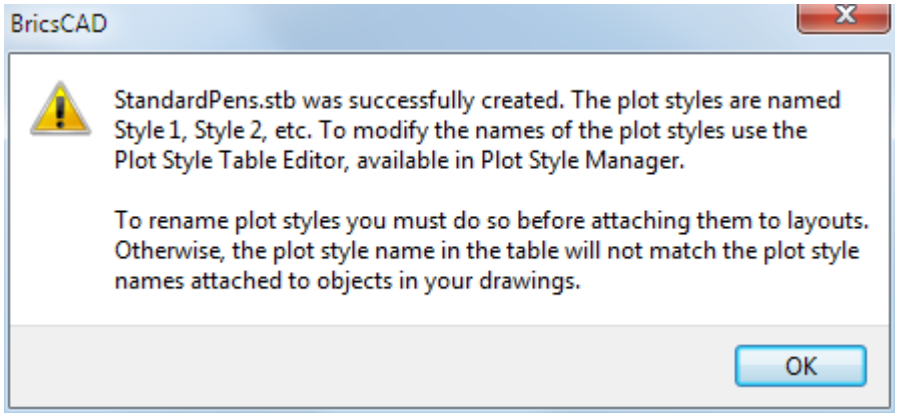
Choose a CTB file, and then click Open.

Displays the next dialog box:



Enter a name for the STB file, and then click Save.

Displays the next dialog box:



Click OK.

Command Options

No options.

Procedures

To convert a Color-dependent plot style tables (CTB) to a Named plot style tables (STB)
To edit plot style tables
To manage plot styles

Related Commands

ConvertPStyles - converts drawings from CTB to STB plot styles.
Layer - assigns plot styles to layers.
PageSetup - assigns plot style to drawings.
PlotStyle - sets the current plot style.
Print - plots drawings with plot styles.
Properties - changes plot styles assigned to entities.
StylesManager - creates and attaches plot styles to drawings.

ConvertOldLights

Converts old light definitions to the current format.

Accessing the Command

command bar: *convertoldlights*

: ***convertoldlights***

The drawing is searched for old light definitions, which are then converted to the current format automatically.

It might be necessary to edit the converted lights to check their settings.

Command Options

No options.

Related Commands

Light - creates a new light in the drawing.
LightList - displays the Drawing Explorer - Lights dialog window.

ConvertOldMaterials

Converts old material definitions to the current format.

Accessing the Command

command bar: *convertoldmaterials*

: ***convertoldmaterials***

The drawing is searched for old material definitions, which are then converted to the current format automatically.

It might be necessary to edit the converted materials to check their settings.

Command Options

No options

Related commands

Materials - creates materials and edits their properties through the Drawing Explorer.

ConvertPoly

Converts lightweight polylines to classic polylines (2D polylines) and vice versa.

Accessing the Command

command bar: *convertpoly*

: ***convertpoly***
Prompts you in the command bar:

Enter polyline conversion option [Heavy/Light] <Light>: (Press Enter for Light or type H for heavy)

Select objects: (Select one or more polylines in the drawing.)
Entities in set: 2

Select objects: (Select more polylines or press Enter to convert the selection.)
2 polyline objects converted.

Command Options

Option	Description
Light	Converts lightweight polylines to classic polylines (2D polylines).
Heavy	Converts classic polylines (2D polylines) to lightweight polylines.

NOTES

- Curve-fit and splined polylines and polylines that have extended object data stored on their vertices cannot be converted to lightweight polylines.
- Editing commands apply to both classic and lightweight polylines.
- The PLINETYPE system variable specifies:
  - Whether classic polylines are converted to lightweight polylines automatically when an older drawing is opened.
  - The polyline type that is created by the Pline command.

Related Commands

Pline - draws polyline lines, arcs, and splines.
Pedit - edits polylines, 3D polylines, and 3D meshes.

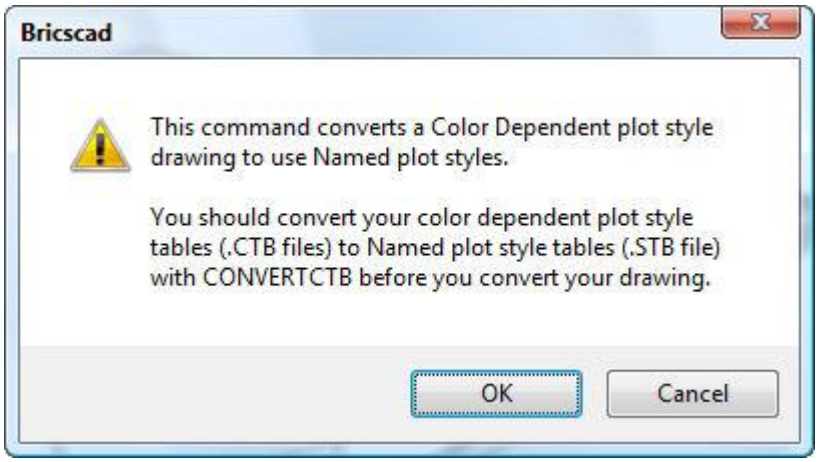
ConvertPStyles

Converts drawings to from CTB (color-based plotting) to STB (plot styles).

Accessing the Command

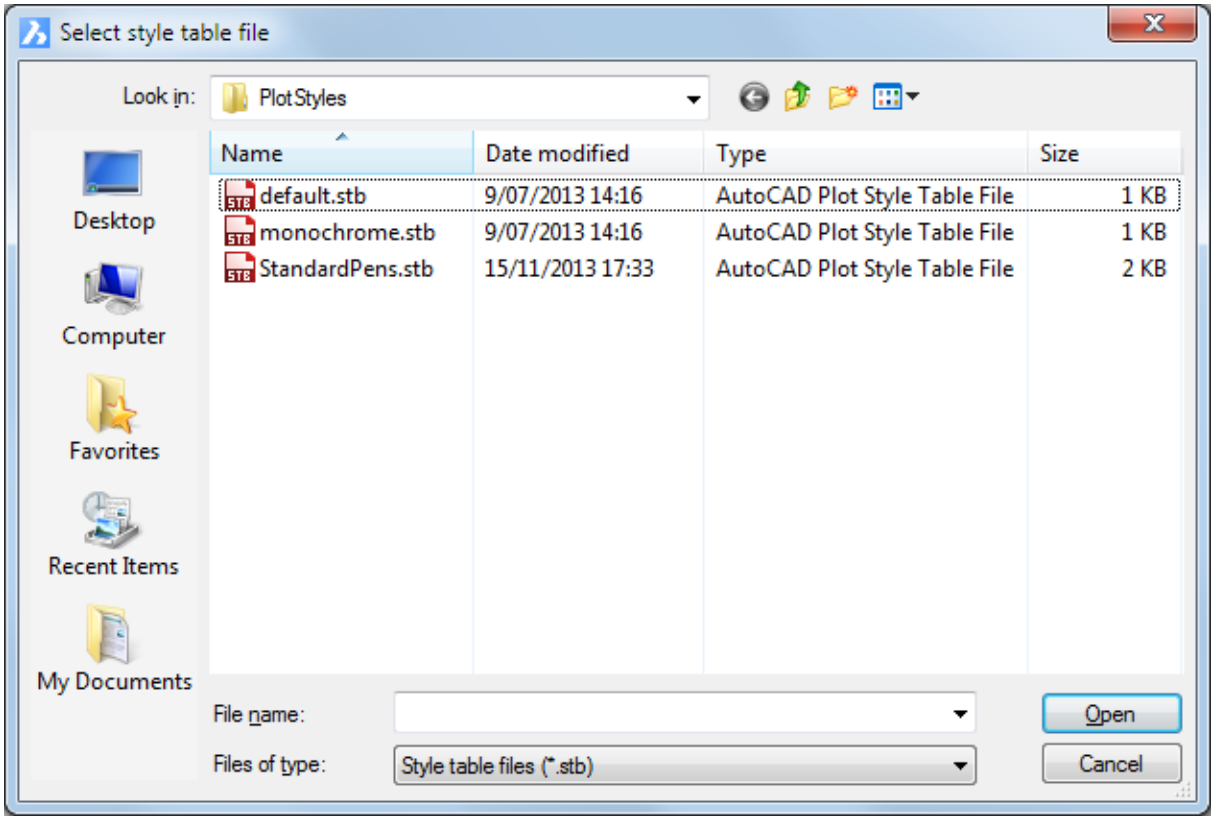
command bar: *convertpstyles*

: ***convertpstyles***
Displays a dialog box:



Click OK.

The next dialog box appears:



Choose an STB file, and then click Open.

BricsCAD reports in the command bar:

Drawing converted from Color Dependent mode to Named plot style mode.

Command Options

No options.

Procedures

To convert a drawing from STB to CTB file type
To convert a drawing from CTB to STB file type
To edit plot style tables
To manage plot styles

Related Commands

ConvertCtb - converts CTB plot style files to STB files.

Layer - assigns plot styles to layers.

PageSetup - assigns plot style to drawings.

PlotStyle - sets the current plot style.
Print - plots drawings with plot styles.
Properties - changes plot styles assigned to entities.
StylesManager - creates and attaches plot styles to drawings.

Copy

Copies entities.

Accessing the Command

command bar: *copy*
aliases: *co, cp*
menu bar: *Modify | Copy*
toolbar: *Modify |*
Quad cursor menu: *Custom |*

: ***copy***
Prompts you in the command bar:

Select entities to copy: (Select one or more entities.)

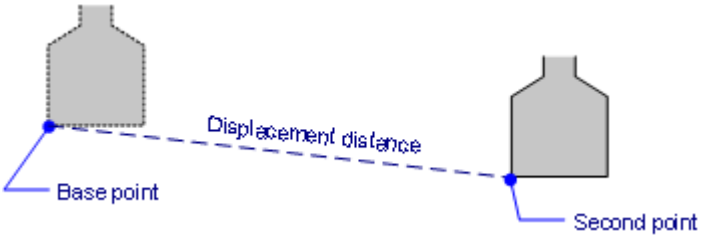
Select entities to copy: (Select more entities, or press Enter to end entity selection.)

Current copy mode: Multiple

Enter base point **[**Displacement**/**mOde**]** **<**Displacement**>:** (Pick a point.)

Enter second point **<**Use base point as displacement**>:** (Pick another point, or press Enter.)

Enter second point **[**Undo**/**Repeat**/**Exit**]:** (Pick another point, or enter an option.)



Command Options

Option	Description
Select Entities to Copy	Use any selection mode to choose the entities to be copied.
Enter Base Point	Specifies the point from which to copy. Use entity snaps to pick the starting point accurately.
mOde	Toggles between single and multiple copying modes; prompts you: **Enter a copy mode [Single/Multiple] <Multiple>:** - enter S or M. • **Single** - makes one copy, and then exits the command. • **Multiple** - repeats the 'Enter second point' prompt until you enter the Exit option.
Displacement	Specifies the distance to position the copy; prompts you: **Enter displacement point <0.0000,0.0000,0.0000>** - enter x,y,z coordinates, or pick a point.
Enter Second Point	Specifies the destination of the copied entities. Hold down the Shift key to copy entities horizontally or vertically. You can use x,y,z coordinates to copy objects in 3D space.
Use Base Point As Displacement	Determines the displacement distance from the coordinates of the base point.

Multiple	(Single mode only.) Switches to multiple-copy mode.
Undo	(Multiple mode only.) Undoes the last copy operation.
Repeat	Repeats the copy using the same displacement; prompts you: Select end point (<ENTER> to key in number of repeats) - specify a point or press Enter. • **Specify a point** - move the cursor in the displacement direction. The further you move away from the source entity, the more copies are created. Click when the desired number of copies displays. • **Enter number of repeats <1>** - press Enter to accept the previous number of repeats or type a number, then press Enter.
Exit	(Multiple mode only.) Exits this command.

Procedures

To copy a selection set
To make multiple copies
To paste a selection set as a block
Using Copy and Paste to copy entities between drawings
Using Copy and Pasteorig to copy entities between drawings
Using Copybase and Paste to copy entities between drawings
Paste a selection as a block in the target drawing
To copy entities using grips

Related Commands

CopyClip - copies selected entities to the Clipboard, which can then be pasted in the same or other drawings.
CopyBase- copies selected entities to the Clipboard with a base point, for pasting at an insertion point.
Array - creates arrayed copies of entities in 2D linear, rectangular, and polar patterns.
3dArray - creates arrayed copies of entities in 3D rectangular and polar patterns.
MInsert - inserts arrayed copies of blocks.
Offset - creates parallel copies of entities.

CopyBase

Copies entities to the Clipboard with a base point.

Accessing the Command

command bar: *copybase*
shortcut: *Ctrl+Shift+C*
menu bar: *Edit | Copy with Base Point*

: ***copybase***

Prompts you in the command bar:
Select base point: (Pick a point, or enter x,y coordinates.)

Select entities to copy to the clipboard: (Choose one or more entities.)

Select entities to copy to the clipboard: (Choose more entities, or press Enter to continue.)

Use the PasteBlock command to paste the copied entities into the same or another drawing.

Command Options

Option	Description
Select base point	Specifies the base point for the entities being copied to the Clipboard. Pick a point, or enter x,y coordinates. The PasteBlock command uses these coordinates to paste the entities from the Clipboard into a drawing.
Select entities to copy to the clipboard	Chooses the entities to copy to the Clipboard. Use any entity selection method.

Procedures

Using Copybase and Paste to copy entities between drawings
Paste a selection as a block in the target drawing

Related Commands

CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteBlock - pasted entities from the Clipboard as a block at an insertion point.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

CopyClip

Copies entities to the Clipboard.

Accessing the Command

command bar: *copyclip*
shortcut: *Ctrl+C*
menu bar: *Edit | Copy*
toolbar: *Standard |*

: ***copyclip***
Prompts you in the command bar:

Select entities to copy to the clipboard: (Choose one or more entities; press Ctrl+A to select all entities.)

Select entities to copy to the clipboard: (Choose more entities, or press Enter to exit entity selection.)

Command Options

Option	Description
Select entities to copy to the clipboard	Choose the entities to be copied; use any entity selection method. Once in the Clipboard, the entities can be pasted into the same drawing, into other drawings, and into other documents with Ctrl+V.

(*) The *PictureExportScale* user preference sets the resolution of the image when the exported geometry is pasted in a bitmap format, e.g. in a Word document.

Procedures

Using Copy and Paste to copy entities between drawings
Using Copy and Pasteorig to copy entities between drawings

Related Commands

ChSpace - moves entities from paper space to model space and vice versa.
CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteBlock - pasted entities from the Clipboard as a block at an insertion point.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

CopyEData

Copies extended entity data from one entity to others.

Accessing the Command

command bar: *copyedata*
transparent: *'Copyedata*
menu bar: *Tools | Entity Data | Copy Entity Data*
toolbar: *Entity Data |*

: ***copyedata***
Prompts you in the command bar:
Copy Entity Data: Name of application or [?]: (Enter name of application, or type ? to List Application Names.)

Select entity from which to copy Entity Data: (Choose one entity.)

Select entity to which to copy Entity Data: (Choose another entity.)
Select entity to which to copy Entity Data: (Choose another entity, or press Enter to exit the command.)

Command Options

Option	Description
Name of Application	Specifies the name of the application that generated the extended entity data.
?	Lists the names of registered applications.
Select Entity From Which to Copy Entity Data	Selects the entity from which to copy edata. The entity must contain edata, which can be created with the EditEData command.
Select Entity To Which to Copy Entity Data	Specifies the entity to receive the edata.

Related Commands

DelEData - erases extended entity data from entities.
EditEData - creates and edits extended entity data.
MoveEData - moves extended entity data to other entities.
ReAssocApp - associates applications with entities.

CopyHist

Copies the command history text to the Clipboard.

Accessing the Command

command bar: *copyhist*

: ***copyhist***

Displays no prompt in the command bar.

All text in the command bar history is copied to the Clipboard; paste the text into another application with **Ctrl+V**.

Command Options

No options.

Related Commands

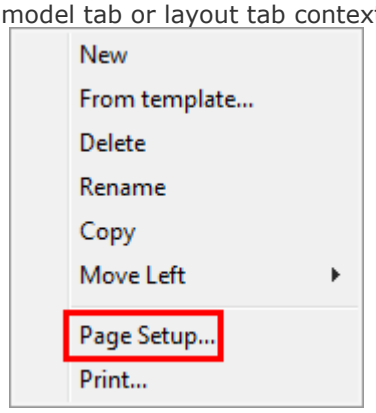
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

CPageSetup

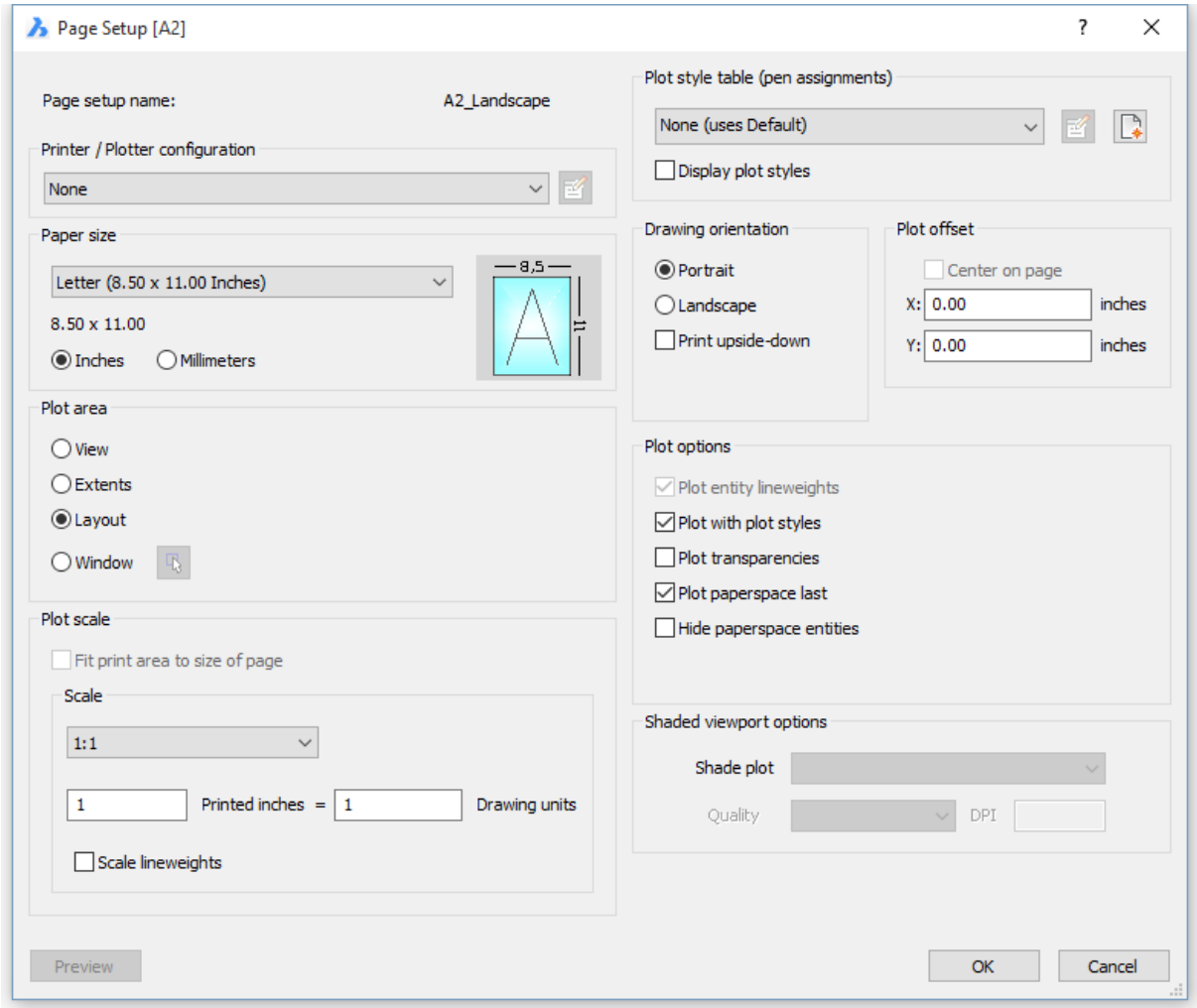
Edits the page setup of the current layout or model space in a dialog box.

Accessing the Command

command bar: *cpagesetup*
model tab or layout tab context menu (right click):



: ***cpagesetup***
Displays a dialog box:



Command Options

See the PageSetup command.

Related Commands

PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.
Print - prints drawings to plotters and files through a dialog box.

CuiLoad

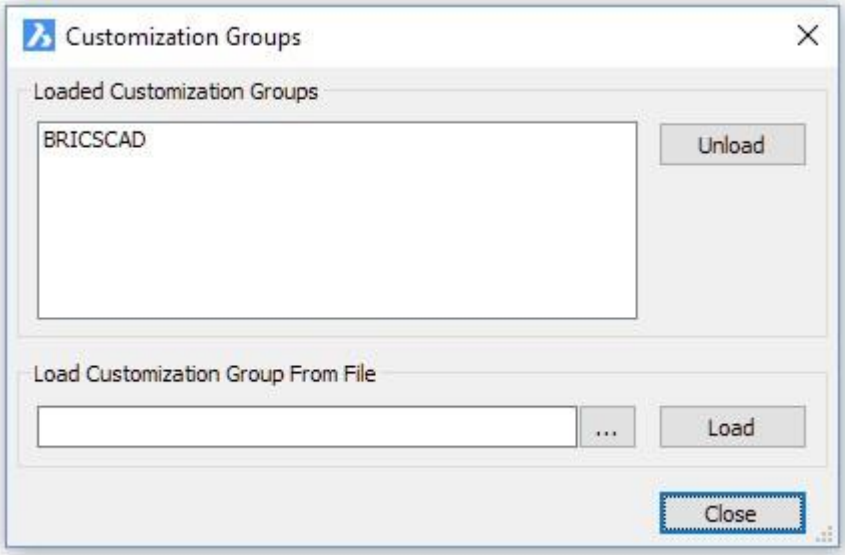
Loads CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files.

Accessing the Command

command bar: *cuiload*
alias: *menuload*

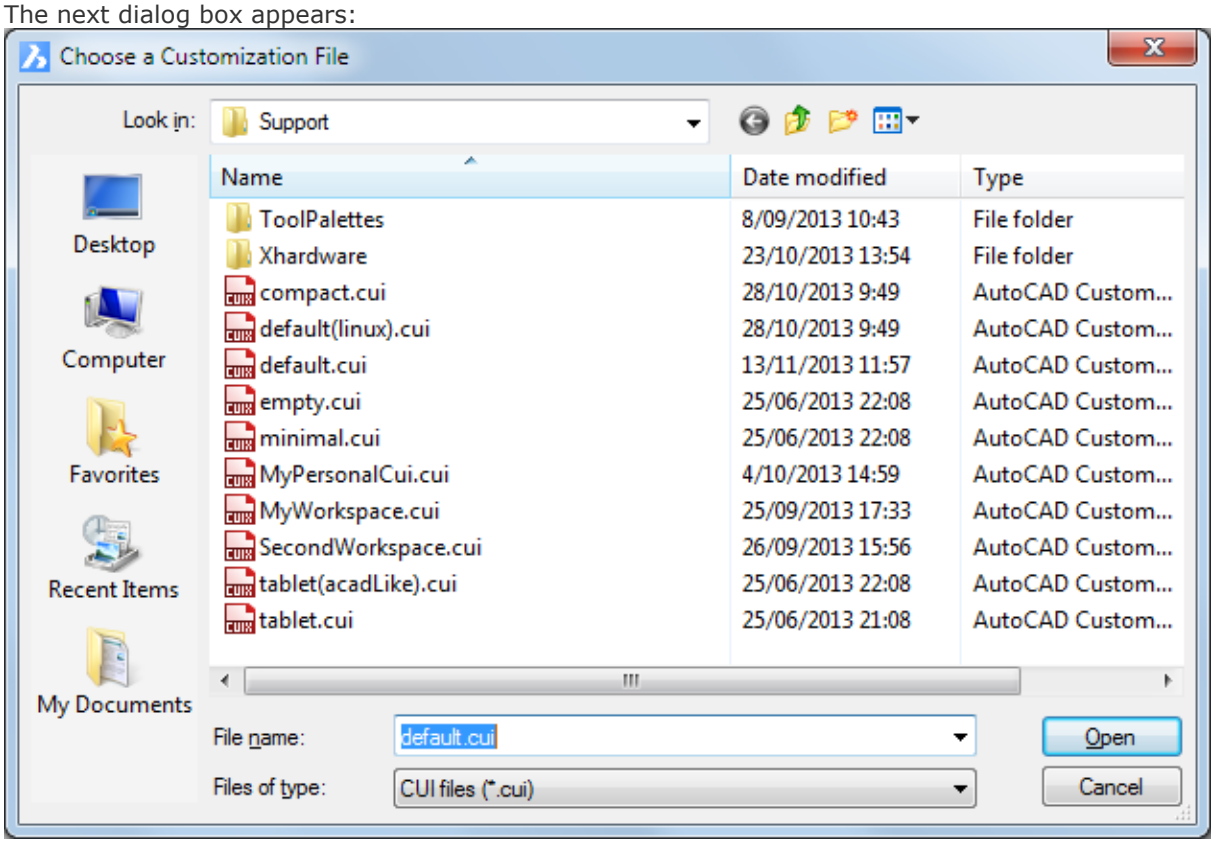
: ***cuiload***

Displays a dialog box:



Click the button.

The next dialog box appears:



Choose a CUI, MNU, MNS, or ICM file, and then click Open.

*The Customization Groups dialog box reappears. Click **Load** to add the customization file to the user interface.*

Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.
...	Displays the Choose a Customization File dialog box.
Load	Loads the opened customization file.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Commands - lists the names of internal and external commands.
CuiUnload - unloads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

CuiUnload

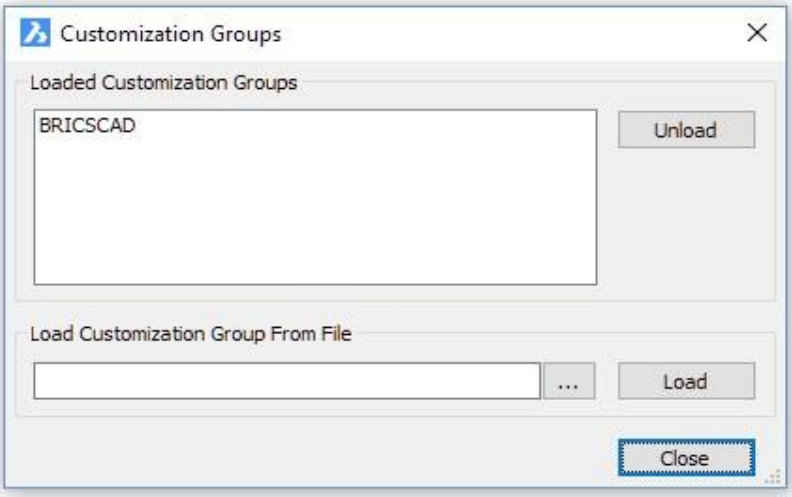
Unloads customization groups from the user interface.

Accessing the Command

command bar: *cuiunload*
alias: *menuunload*

: ***cuiunload***

Displays a dialog box:



Choose a customization group, and then click **Unload**.

Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.
...	Displays the Choose a Customization File dialog box.
Load	Loads the opened customization file.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Commands - lists the names of internal and external commands.
CuiLoad - loads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

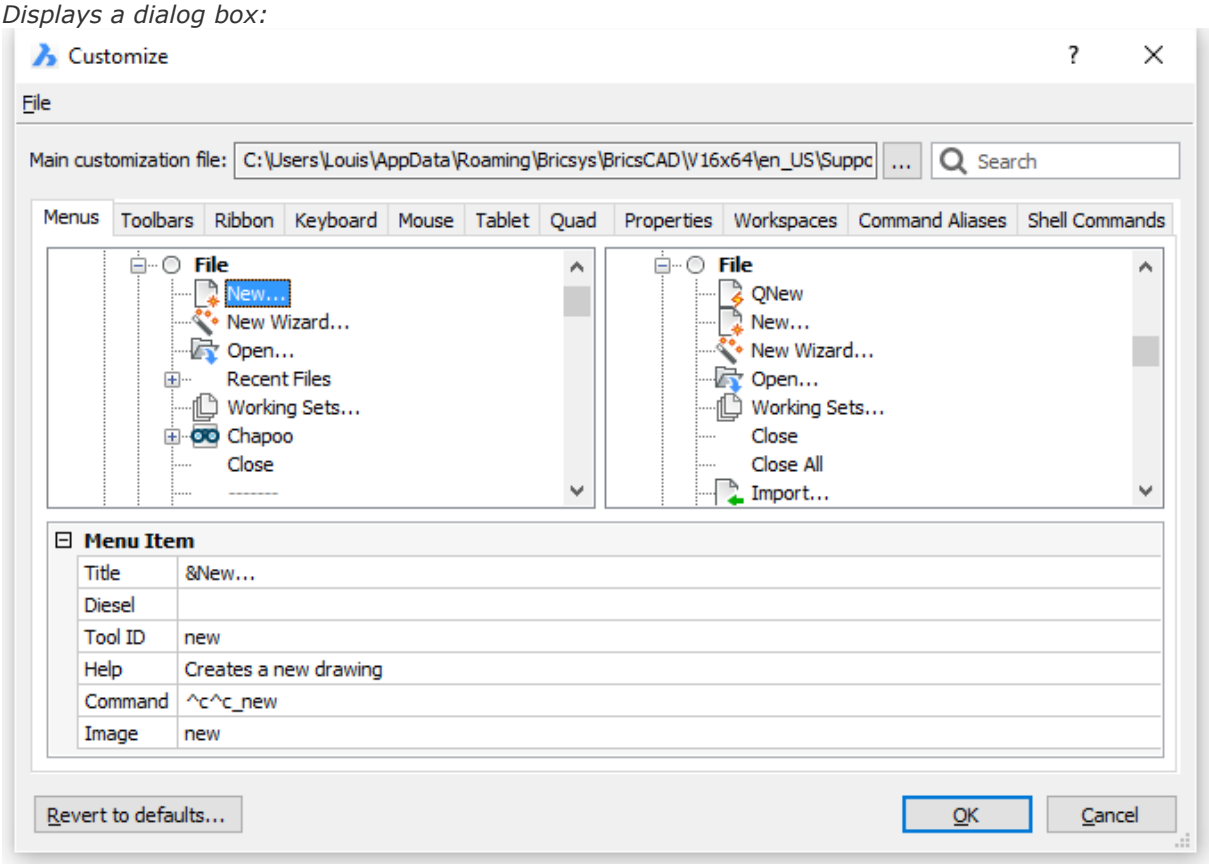
Customize

Customizes the user interface of BricsCAD: Menus, Toolbars, Keyboard shortcuts, Mouse actions, tablet, Quad settings, Properties, Workspace settings, Aliases, and Shell Commands.
For more details on using this command, see also Customizing BricsCAD in the User Guide.

Accessing the Command

command bar: *customize*
alias: *cui*
menu bar: *Tools | Customize*

: ***customize***
Displays a dialog box:

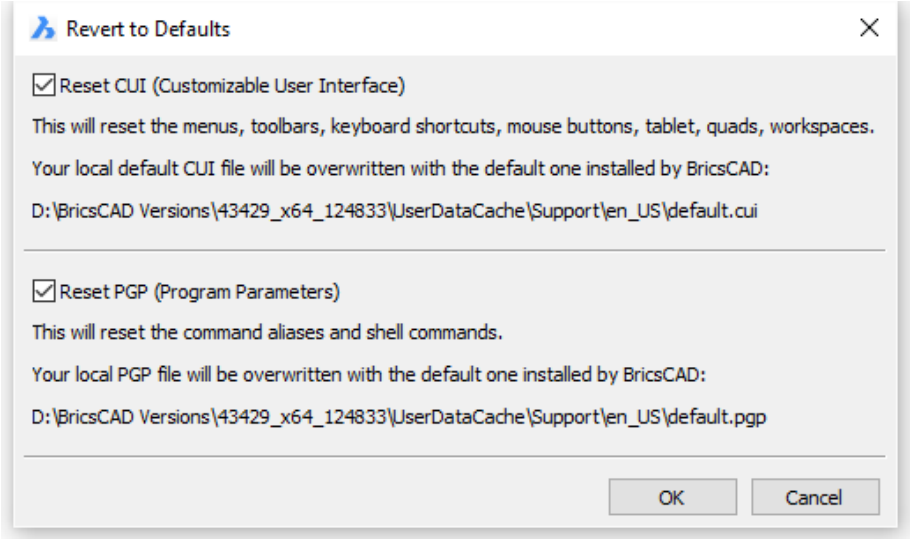
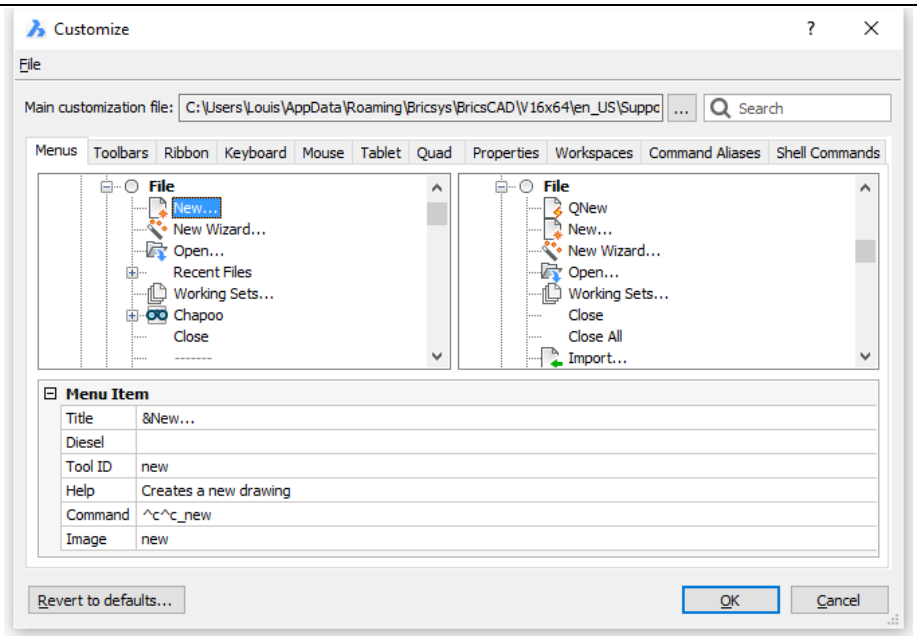


Create or modify a user interface elements, and then click OK.

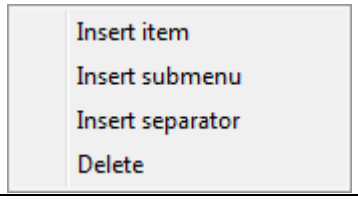
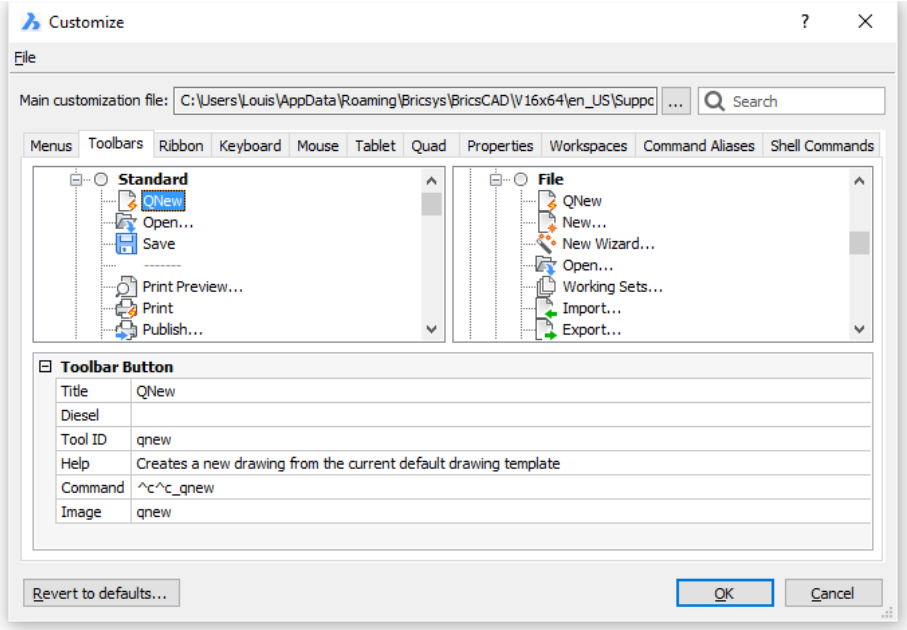
For details on using this dialog box, see the "Customizing BricsCAD" ebook.

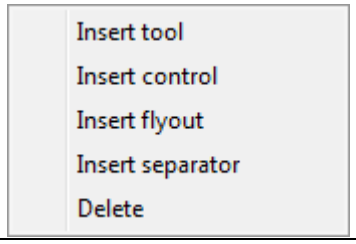
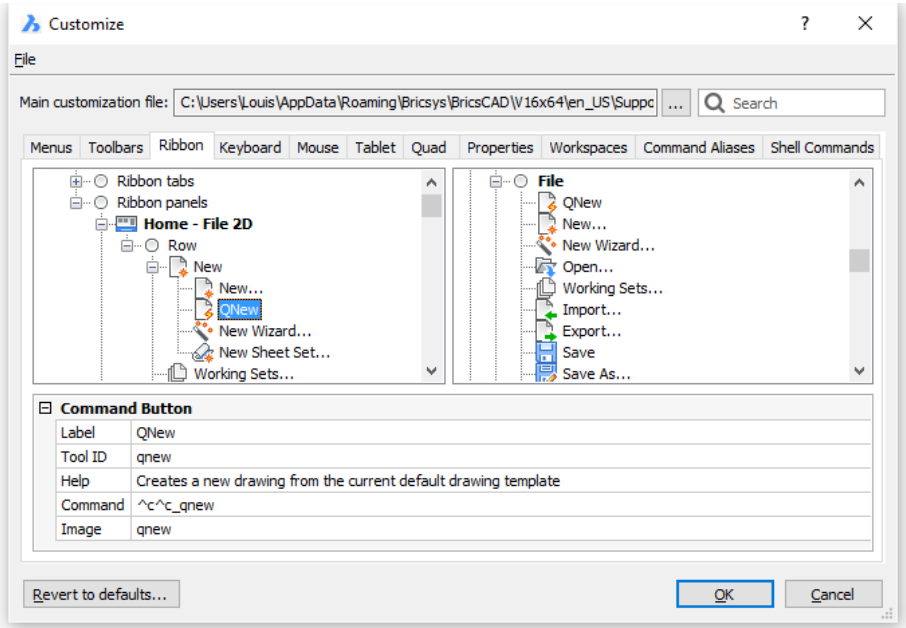
Command Options

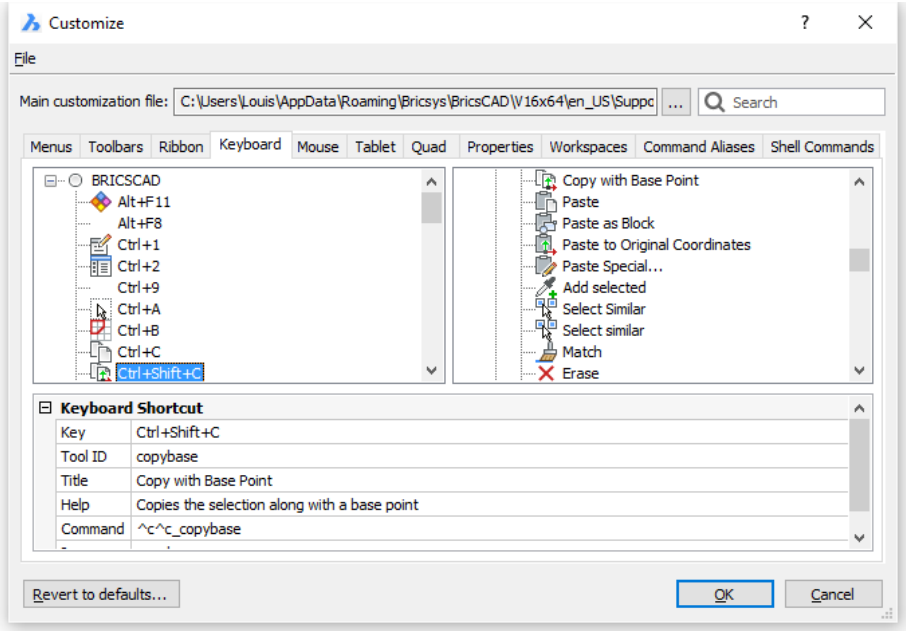
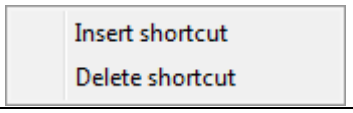
Options	Description
Main Customization File	Specifies the CUI file that defines the menu, toolbar, and keyboard customization. Aliases and shell commands are defined in a PGP file.

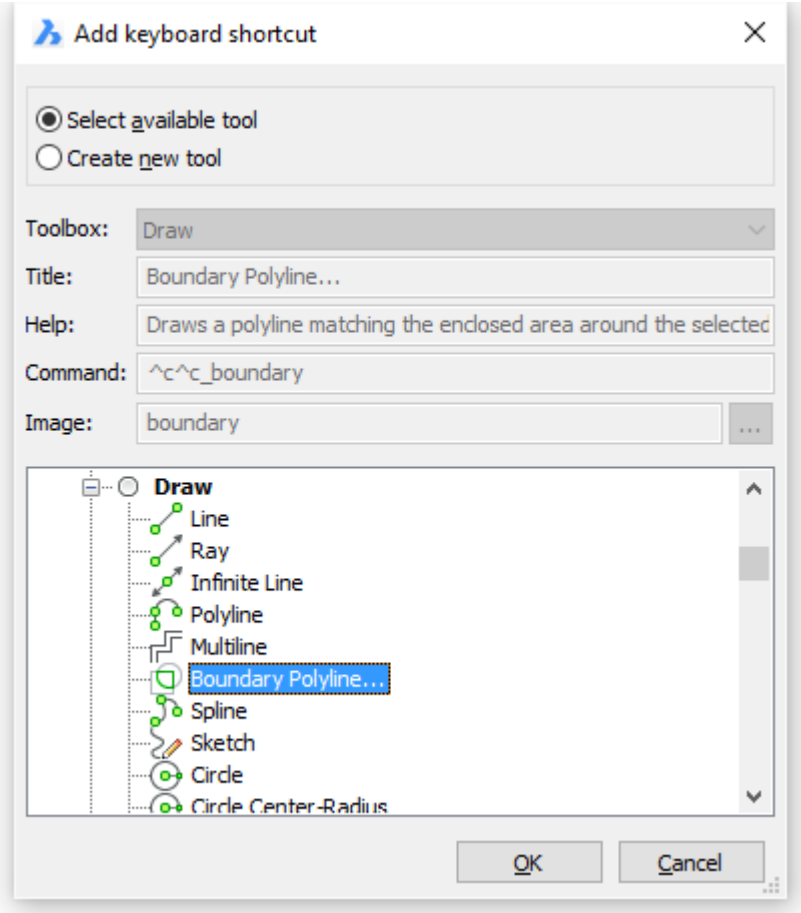
Revert to Defaults	Returns all user interface elements to their out-of-the-box state. Displays the *Revert to Defaults* dialog box:  This button is handy when you make a big mistake with customization.
Reset toolbars, menus, and keyboard shortcuts	Overwrites the default.cui file the default installed originally by BricsCAD.
Reset aliases and shell commands	Overwrites the default.pgp file the default installed originally by BricsCAD.
Menus Tab	

	• To add a new menu item: Select a command in the right and pane, then drag the command to the menu in the left hand pane. • To move a menu item: Drag the item to its new location.

Shortcut Menu	*Right-click a menu item:* 
Insert Item	Adds an item to the menu.
Insert Submenu	Adds a submenu to the menu.
Insert Separator	Inserts a separator line between menu items.
Delete	Erases the selected menu item.
Menu Item	
Title	Specifies the name displayed by the menu item. Prefix a letter with & to create an Alt-key shortcut, such as &Line.
Diesel	Specifies the Diesel code to be employed by the menu item.
Tool ID	Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.)
Help	Specifies the help string displayed on the status bar.
Command	Specifies the command(s) or macros.
Image	Specifies the image to display next to the menu item; click ... to access the file dialog box.
Toolbars Tab	 • To add a new toolbar item: Select a command in the right and pane, then drag the command to the toolbar in the left hand pane. • To move a toolbar item: Drag the item to its new location.
Shortcut Menu	Right-click a toolbar item:

	
Insert Tool	Inserts a new button in the selected toolbar.
Insert Control	Inserts a control (droplist).
Insert Flyout	Inserts a flyout (sub-toolbar).
Insert Separator	Inserts a separator line.
Delete	Erases the selected toolbar button.
Toolbar Button	
Title	Names the toolbar button; displayed by the tooltip.
Diesel	Specifies the Diesel code to be employed with the button.
Tool ID	Identifies the toolbar item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.)
Help	Specifies the help string displayed on the status bar.
Command	Specifies the command(s) or macros.
Image	Specifies the image to use for the toolbar button; click ... to access the file dialog box.
Ribbon Tab	
Ribbon tabs	To add a new ribbon tab: • Right click the *Ribbon tabs* node and click *Append ribbon tab* in the context menu to add a new tab at the bottom of the list. • Right click an existing tab and choose *Insert ribbon tab* in the context menu to add a new tab above the selected tab.

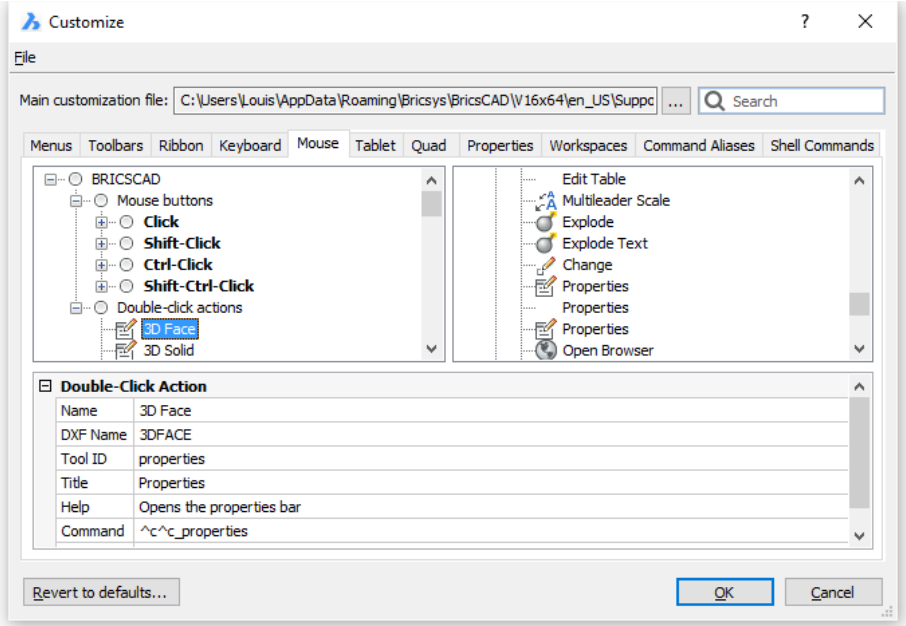
	Drag a tab to modify the order. To add a panel to a tab: • Right click the tab name and choose *Append panel* in the context menu to add a panel at the bottom of the list. • Right click an existing panel and choose *Insert panel* in the context menu to ad a panel above the selected panel. Drag a panel to modify the order.
Ribbon panels	See the Ribbon item in the BricsCAD User Guide.
Keyboard Tab	 To assign a different command to a keyboard shortcut, select a command in the right and pane, then drag the command to the keyboard shortcut in the left hand pane.
Shortcut Menu	Right-click a keyboard shortcut: 
Insert Shortcut	Adds a new keyboard shortcut to the list; displays dialog box:



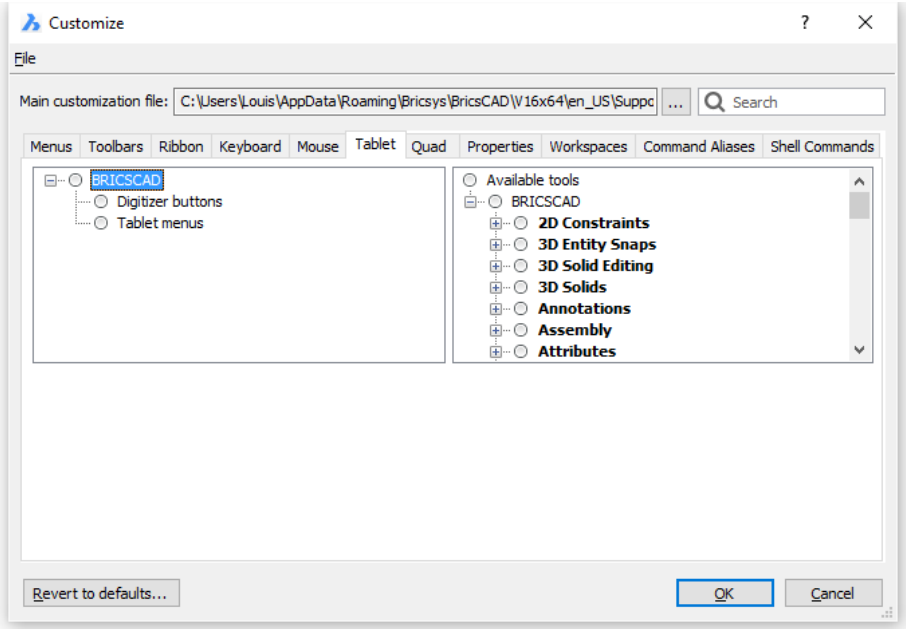
Select Available Tool	Assigns an existing command to the keyboard shortcut.
Create New Tool	Assigns a new command/macro to the shortcut.
Toolbox	Specifies the toolbox to which to add the new command; click the droplist, and then choose a toolbox.
Title	Names the new tool.
Help	Specifies the help text.
Comman d	Specifies the command(s) or macros.
Image	Specifies the name of the image file; click ... to access the file dialog box.
Delete Shortcut	Erases the selected shortcut.
Keyboard Shortcut	
Key	Specifies the keyboard shortcut.
Tool ID	Identifies the keyboard shortcut to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.)
Help	Specifies the text of the help string, even though help strings are not displayed by keyboard shortcuts.

Command	Specifies the command(s) to associate with the keyboard shortcut.
Image	Specifies the image to associate with the command, even though images are not used with keyboard shortcuts.

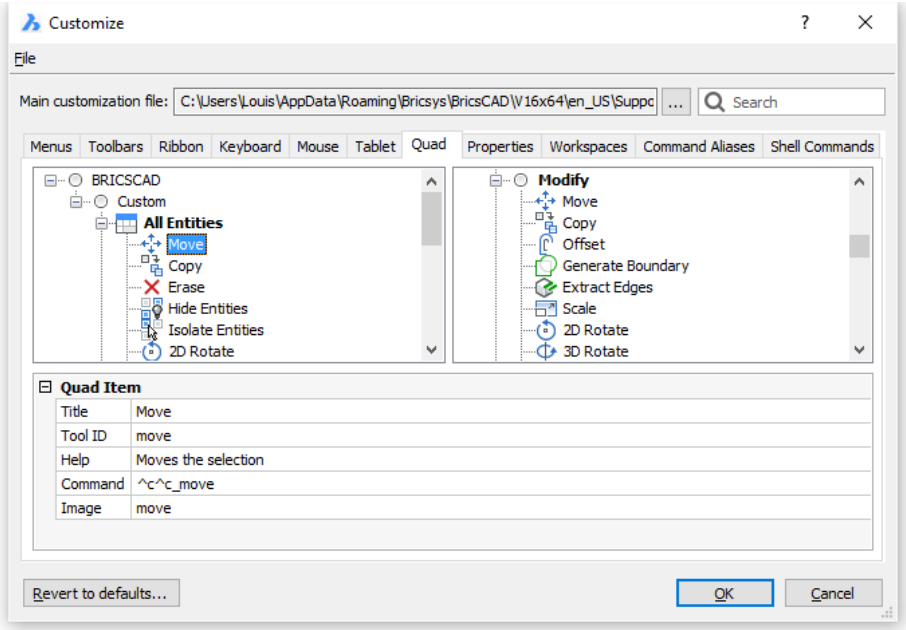
Mouse Tab



Tablet Tab



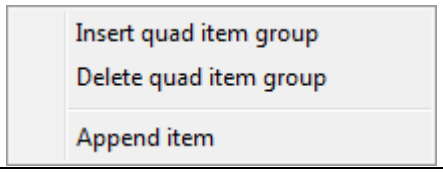
Quad Tab



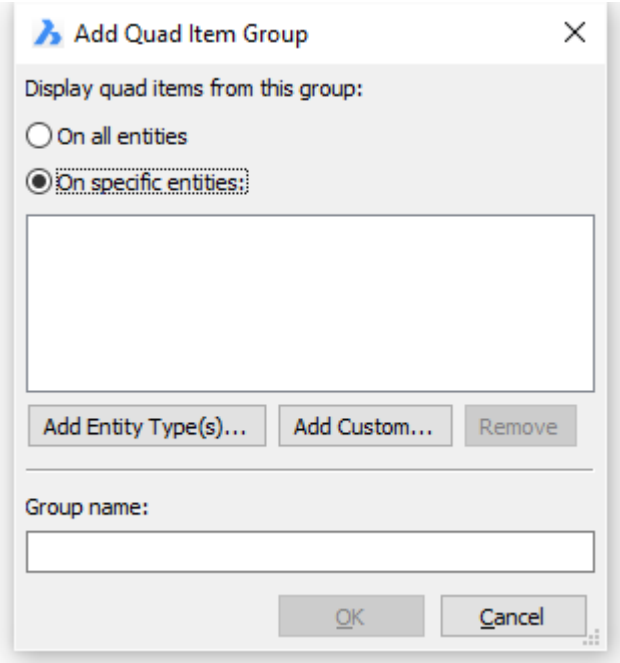
- To add a command to a Quad cursor menu group:
  Select a command in the right and pane, then drag the command to the Quad menu group in the left hand pane.
- To move a command:
  Drag the command to its new location.

Shortcut Menu

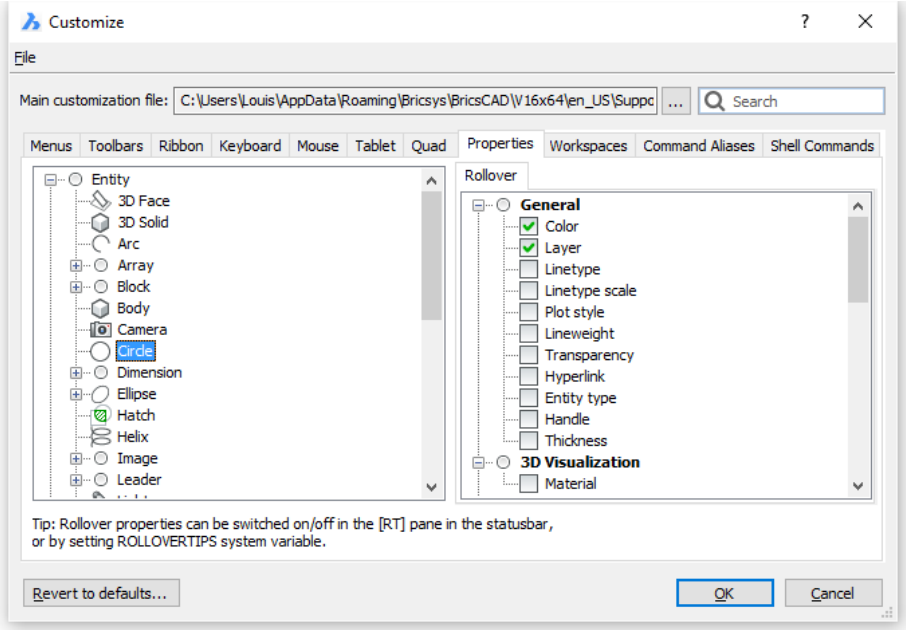
Right-click a keyboard shortcut:

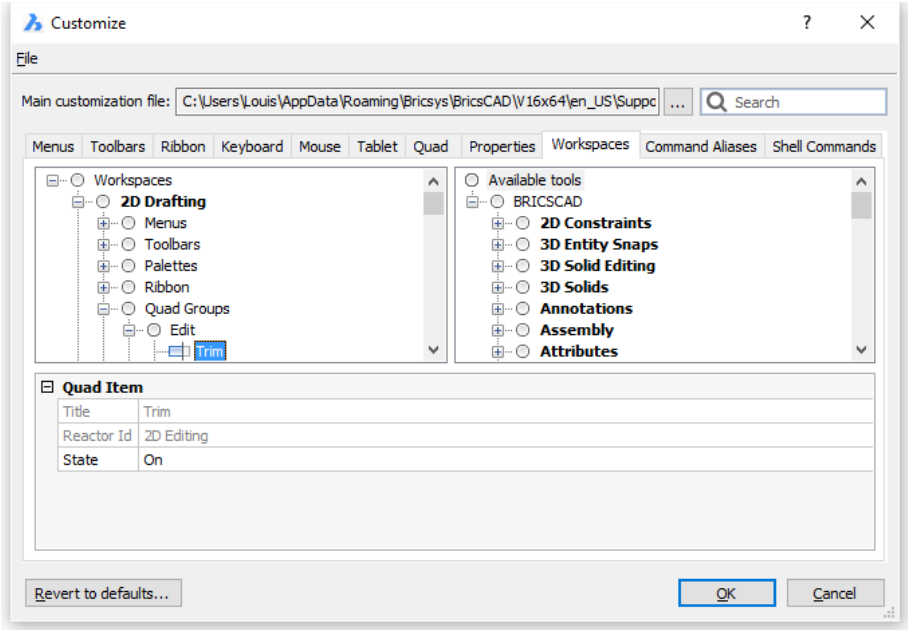


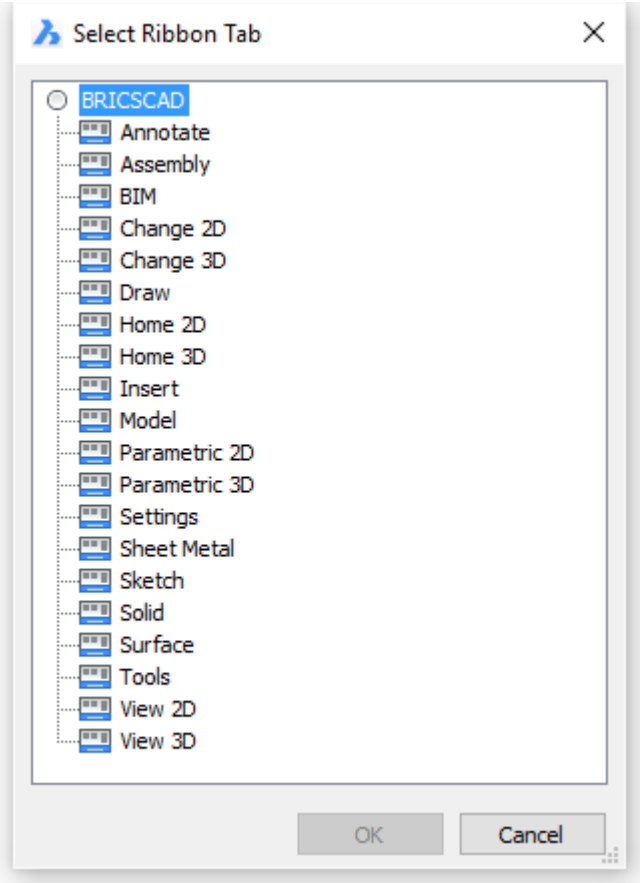
Insert quad item group

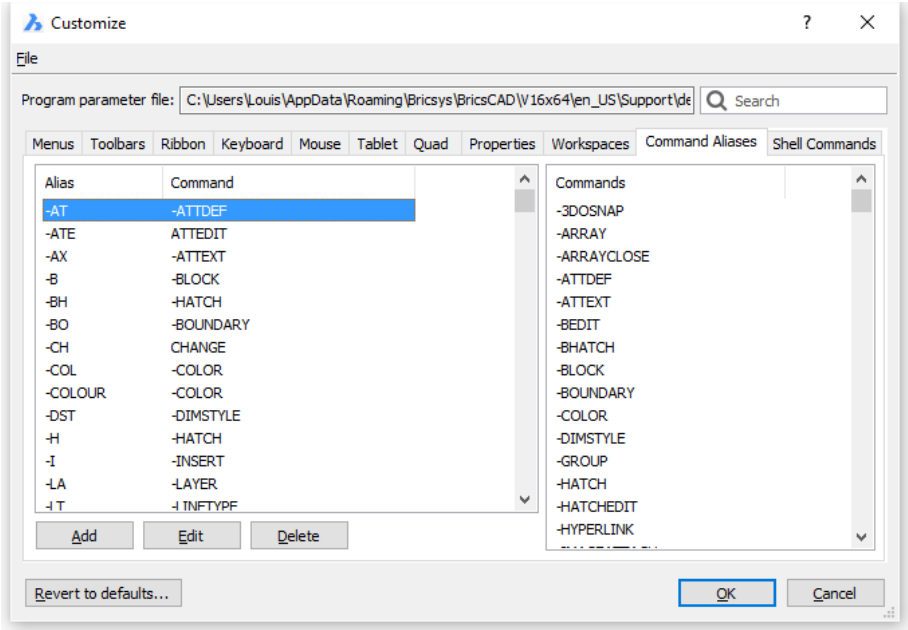
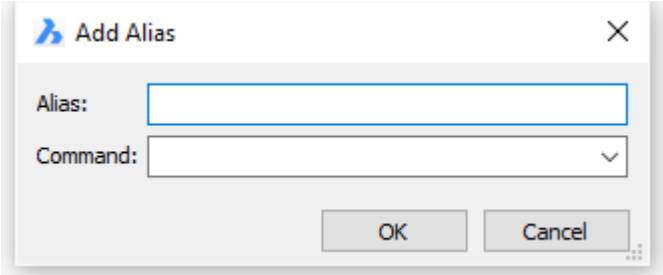


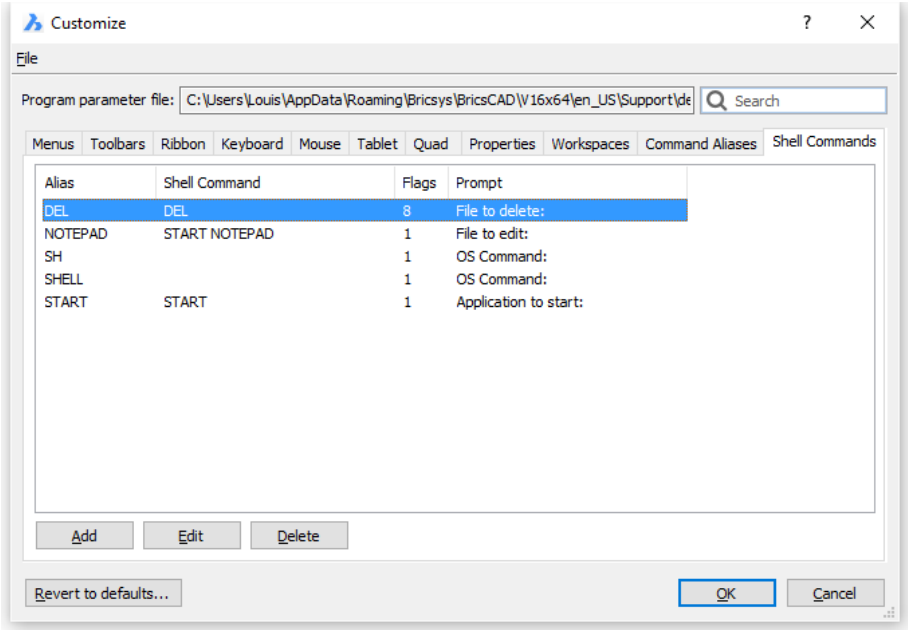
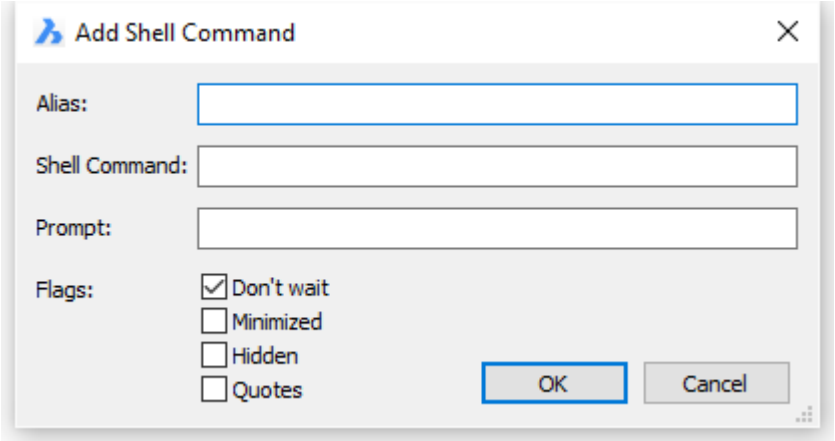
- On all entities: the Quad item group is available for all entity types.

	• On specific entities: the Quad item group is available for one or more entity types only. Click the *Add Entity Type(s)...* button to specify one or more default entity types. Click the *Add Custom...* button to add custom entity types. • Group name: type a name in the name field.
Delete quad item group	Select the Quad item group you want to delete.
Append item	Adds a new menu item.
Properties Tab	 To specify which properties show when the ROLLOVERTIPS system variable is ON, select an entity in the left hand pane, then check the desired properties in the right hand pane.

Workspace Tab	
Menus	Controls the display of menus in the menu bar. • To modify the order: drag the menu. • To control the display of a menu: click the *State* field and select *On* or *Off*.
Toolbars	Lists the toolbars which are currently open in the workspace.
Palettes	Controls which tool panels are open in the workspace. Palettes ☑ BIM Compositions ☑ Command Line ☐ Render Materials ☐ Mechanical Browser ☑ Properties ☐ Ribbon ☐ Sheet Sets ☐ Tool Palettes ☐ Layers ☐ Model Browser
Ribbon	Controls which ribbon tabs are available in the workspace. Ribbon Home View Settings Tools Insert Draw Model BIM Change Parametric • Drag a tab to modify the order.

	• Right click a tab in the list, then choose *Insert tab* in the context menu. 
Quad Groups	Controls the display and the order of the Quad item groups. • To modify the order: drag the Quad item groups. • To control the display of a Quad item group: click the *State* field and select *On* or *Off*.
On Switch	Sets the commands to be executed when switching to the workspace. To ad a command, do one of the following: • Right click, then choose *Append Tool* in the context menu. • Drag a command from the right hand pane to the *On Switch* command list.
Context Menu	Insert workspace Insert separator Delete
Insert workspace	Creates a new workspace.
Insert separator	Inserts a separator before the selected workspace.
Delete	Deletes the selected workspace.

Aliases Tab	
Add	Adds an alias; displays a dialog box: 
Alias	Specifies the name of the alias, a command abbreviation.
Command	Specifies the command to which the alias is assigned.
Edit	Edits the selected alias.
Delete	Erases the selected alias.

Shell Commands Tab	
Add	Adds a shell command; displays the following dialog box:  Shell commands run programs external to BricsCAD.
Alias	Specifies the name of the shell command; this is the command you enter at the ' : ' prompt.
Shell Command	Specifies the operating system command to execute, or program to run. To run a program, use the Start prefix, as in Start Notepad.exe.
Prompt	Specifies the prompt to display in the command bar, such as "Enter file to edit:"
Flags	Determines how the shell command operates: Don't Wait - BricsCAD immediately returns to the command prompt without waiting for the application to finish. Minimized - the application runs minimized. Hidden - the application does not appear on the screen. Quotes - the shell command uses quotations when the command string uses spaces, such as "C:\cad programs\file name.exe".

Edit	Edits the selected shell command.
Delete	Removes the selected shell command.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Commands - lists the names of internal and external commands.
CuiLoad and CuiUnload - loads and unloads menu files.
MenuLoad and MenuUnload - loads and unloads menu files.
Netload - loads a .NET application.
ProfileManager - allows to set current, create, copy, delete, import and export user profiles through a dialog box.
ToolPalettes - opens the Tool Palettes bar.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.
Workspace - sets the current workspace and lets you create, modify and save workspaces.
WsSettings - opens the Customize Workspaces dialog box.

CutClip

Copies entities to the Clipboard, and then erases them from the drawing.

Accessing the Command

command bar: *cutclip*
shortcut: *Ctrl+X*
menu bar: *Edit | Cut*
toolbar: *Standard |*

: ***cutclip***
Prompts you in the command bar:

Select entities to cut to the clipboard: (Choose one or more entities; press Ctrl+A to select all entities.)

Select entities to cut to the clipboard: (Choose more entities, or press Enter to exit entity selection.)

Command Options

Option	Description
Select entities to cut to the clipboard	Choose the entities to be cut; use any entity selection method. Once in the Clipboard, the entities can be pasted into the same drawing, into other drawings, and into other documents with Ctrl+V.

(*) The *PictureExportScale* user preference sets the resolution of the image when the exported geometry is pasted in a bitmap format, e.g. in a Word document.

Procedures

Using Cut and Paste to move entities between drawings
Move a selection as a block

Related Commands

ChSpace - moves entities from paper space to model space and vice versa.
CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
PasteBlock - pasted entities from the Clipboard as a block at an insertion point.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

Cylinder

Creates a cylinder as a polygon mesh (Classic); creates a cylinder as a 3D solid (Pro and Platinum).

Accessing the Command

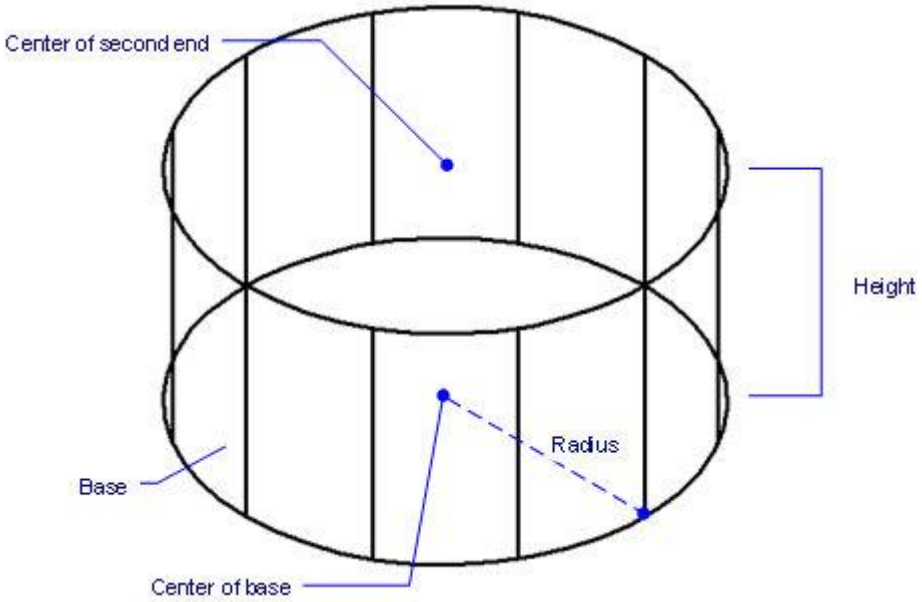
command bar: *cylinder*
alias: *cyl*
menu bar: *Model | 3D Solids | Cylinder*
toolbar: *3D Solids |*

: ***cylinder***
Prompts you in the command bar:

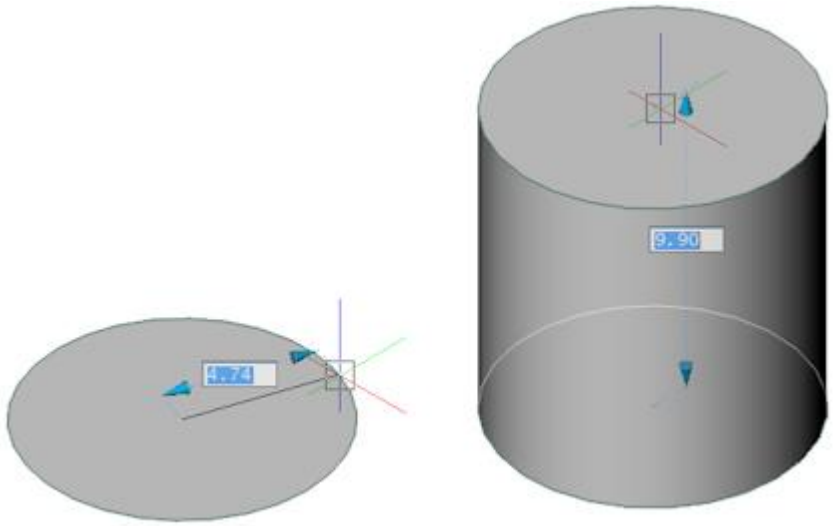
Select center point or [3Point/2Point/Tangent-tangent-radius/Elliptical]: (Enter coordinates, pick a point, or choose an option.)

Radius of base of cylinder or Diameter <x.xx>: (Enter a distance or press Enter to accept the previously entered value, pick a point, or enter D.)

Specify height or [2Point/Axis endpoint] <x.xx>: (Enter a distance or press Enter to accept the previously entered value, pick a point, or choose an option.)



If Dynamic Dimensions are active, you can enter the dimensions in the dynamic dimensions entry fields.



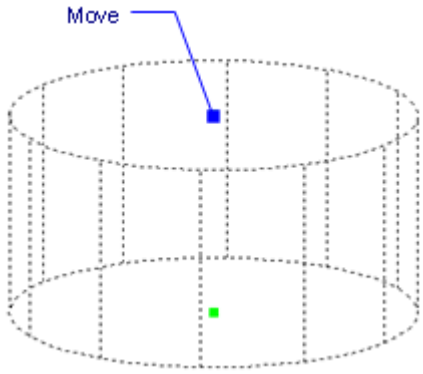
Command Options

Option	Description
Diameter	Specifies the radius of the cylinder; prompts you: Diameter of base of cylinder <x.xx>: - enter coordinates, a distance, pick a point or press Enter to accept the previously entered value.
2Point	Specifies the height of the cylinder as the distance between two points. Prompts you: Specify first point - enter coordinates or pick a point. Specify second point - enter coordinates or pick a point.
Axis endpoint	Specifies the endpoint of the axis, allowing to create a cylinder of which the axis is not parallel to the Z-axis of the current coordinate system. Prompts you: Specify axis endpoint - enter coordinates or pick a point.

Grips Editing

Cylinders can be edited directly through grips:

1. Select the cylinder. Notice that it has two grips.
2. Drag a grip to move the cylinder. (Both grips perform the same function: move.)


Move

Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Region - creates 2D regions from 2D entities.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

D

-DesignTableEdit

Allows to edit designtables.

Accessing the Command

command bar: *-designtableedit*

: ***-designtableedit***
prompts you in the command bar:
Design table [eXport/Replace/Delete/Configurations/Parameters/Apply configuration/? to list] <? to list>: *(Choose an option.)*

Command Options

Option	Description
Export	Exports the content of a design table entity in .csv format file; prompts you: Enter design table name <DesignTable_1> - Type the name of the design table you want to export or press Enter to accept the default name. The *Select design table* file dialog box displays. Do the following: • Type a name in the *Name* field. • Select a folder. • Click the *Save* button.
Replace	Replaces the content of a design table with data from a .csv format file; prompts you: Enter design table name <DesignTable_1> - Type the name of the design table you want to replace or press Enter to accept the default name. The *Select design table* file dialog box displays. Do the following: • Select a .csv design table file. • Click the *Open* button.
Delete	Deletes a Design Table; prompts you: Enter design table name <DesignTable_1> - Type the name of the design table you want to delete or press Enter to accept the default name.

Configurations	Allows to create, remove or view configurations; prompts you:
	Configuration [Save current/Remove/? to list] <? to list> - choose an option.
	• Save current : Creates a new configuration as a copy of an existing one.
	Prompts you:
	  • Enter design table name <v1_2> - type the name of an existing table.
	  • Enter new configuration name <v1_2_1> - type a name for the new table or press Enter to accept the default name.
	• Remove: Removes a configuration.
	Prompts you: Enter design table name <v1_2> - type the name of the table to remove.

	• ? to list:
Parameters	Allows to manage parameters.
Apply Configuration	Allows to create a new row using the current values of variables.
? to list	Lists design table data in the Prompt History dialog box.

Related Commands

-BmParameters - allows to list and edit the parameters of inserted components and associative array
-Parameters - manages dimensional constraint parameters in the current drawing.
DesignTable - creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser and parametric component properties.
MechanicalBrowserOpen - opens the Mechanical Browser panel.

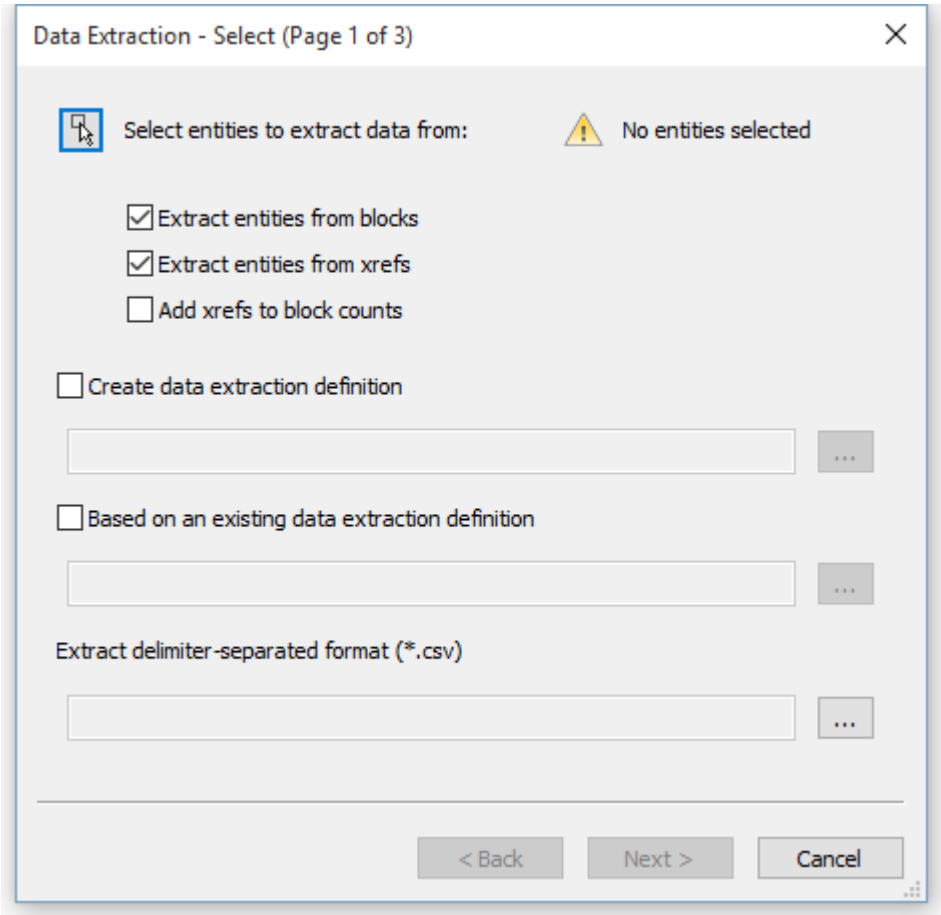
DataExtraction

Exports entity properties, block attributes and drawing information to a an external file.

Accessing the Command

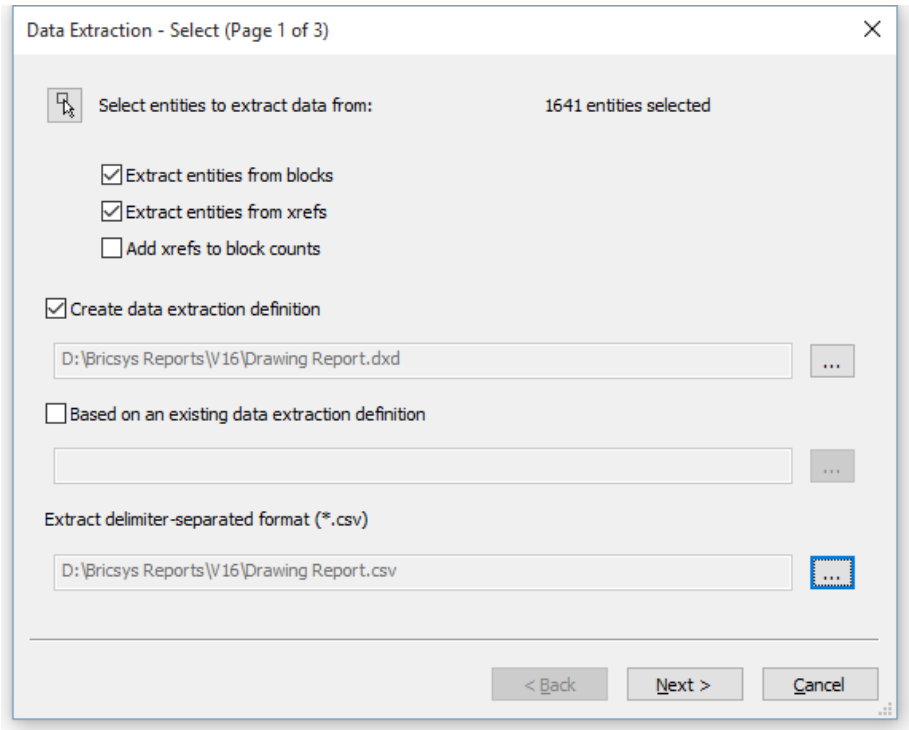
command bar: *dataextraction*
menu bar: *Tools | Data Extraction*

: ***dataextraction***
Displays a dialog box:

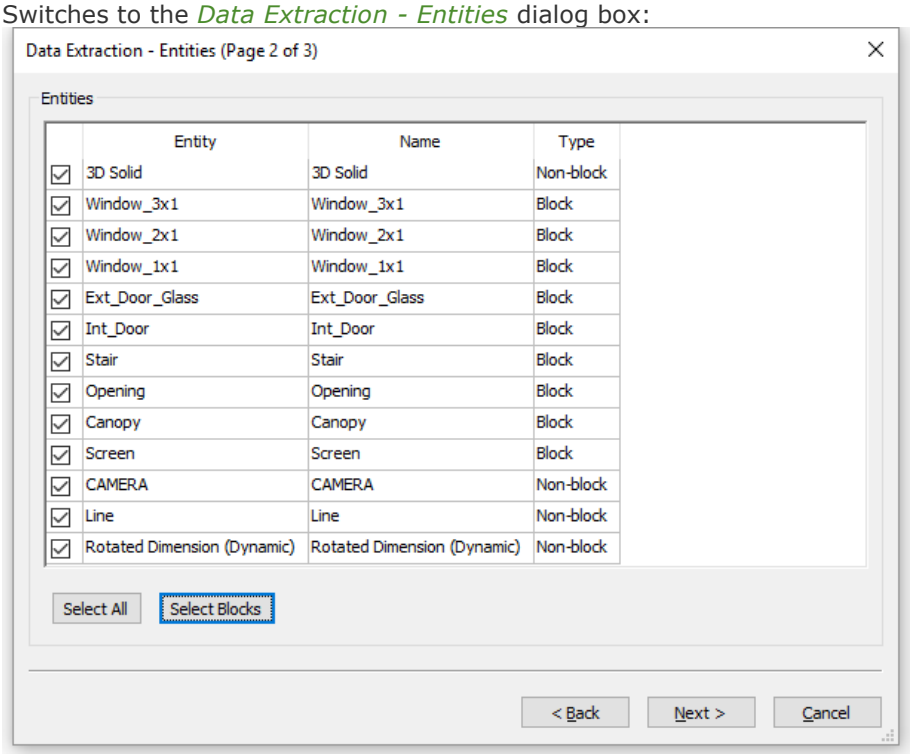


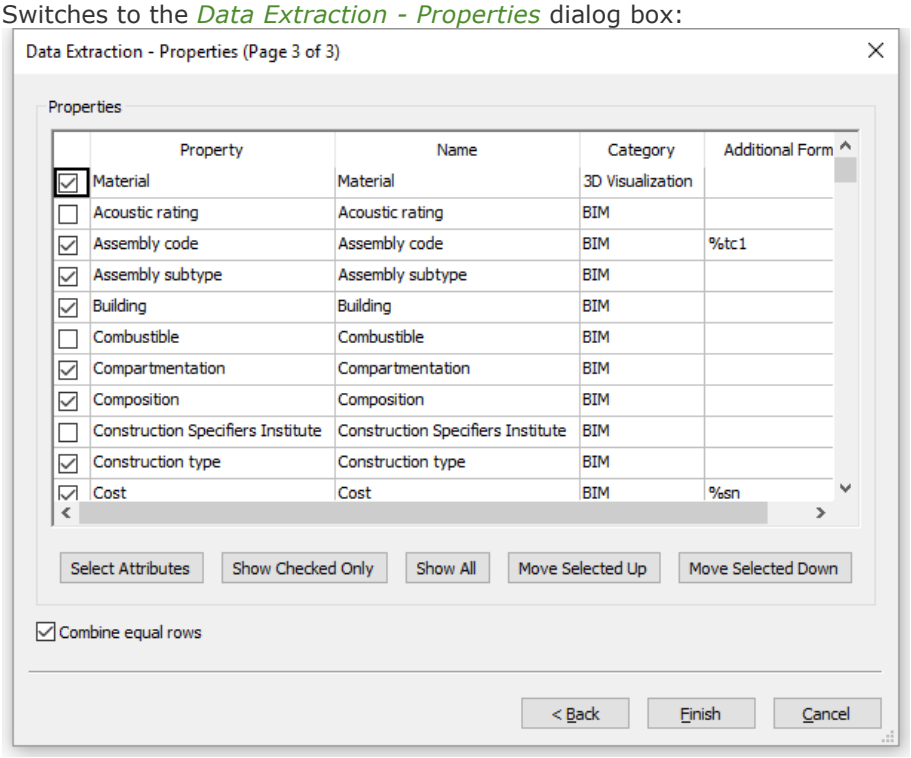
Command Options

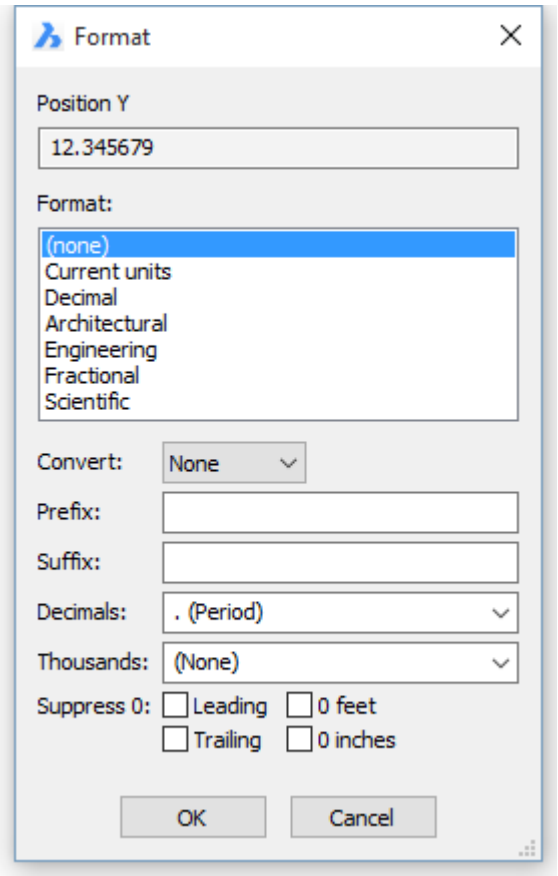
Option	Description
Select entities to extract data from:	Click the *Select* button ( ). The dialog box temporarily closes to let you select entities in the drawing. Press Enter or right click to stop selecting entities. No entities selected is replaced by 'xxx entities selected'



Extract entities from blockx	Include entities in selected blocks.
Extract entities from Xrefs	Include entities in selected Xrefs.
Add Xrefs to block counts	Count selected Xrefs as blocks.
Create data extraction definition	Creates a new *Data Extraction Definition* file (*.dxd). • Check the option, then click the *Browse* button (...). A *File* dialog displays. • Select a folder. • Type a name in the *File name* field, then click the *Save* button. The *File* dialog closes.
Based on an existing data extraction definition	Uses an existing *Data Extraction Definition* file as a template to create the new one; else the new *.dxd file is created from scratch. • Check the option, then click the *Browse* button (...). A *File* dialog displays. • Select a *.dxd file, then click the *Open* button. The *File* dialog closes.
Extract delimiter separated format (*.csv)	Defines the export file. • Click the *Browse* button (...). A *File* dialog displays. • Type a name in the *File name* field or select an existing *.csv file, which then will be overwritten. • Click the *Save* button.

Next	The *Next* button is disabled until the *Data Extraction Definition* (*.dxd) and *.csv files are properly defined. Switches to the *Data Extraction - Entities* dialog box: 
	Indicates whether an item is selected (☑) or not (☐). Select an item, then click the checkbox to check/uncheck the item. Select multiple items: • Select an item, the press and hold the Shift-key and click another item to select a series of items. • Pres and hold the Ctrl-key to select multiple entities. Click again, while holding the Ctrl-key pressed to unselect. Click the checkbox to check/uncheck the selection. Click the empty tile in the column heading to sort the list. Click again to sort in reverse order.
Entity	Displays a list of entity types and blocks in the selection according to the options set in the *Data Extraction* dialog. Click the *Entity* column heading to sort the list; click again to sort in reverse order.
Name	Displays a list of entity names and block names. Click the *Name* column heading to sort the list; click again to sort in reverse order.
Type	Indicates the type of the selected entity: Non-Block, Block or Block with attributes. Click the *Type* column heading to sort the list; click again to sort in reverse order.
Back	Switches back to the *Data Extraction* dialog box.

Next	Switches to the *Data Extraction - Properties* dialog box: 
☑	Indicates whether an item is selected (☑) or not (☐). Select an item, then click the checkbox to check/uncheck the item. Select multiple items: • Select an item, the press and hold the Shift-key and click another item to select a series of items. • Pres and hold the Ctrl-key to select multiple entities. Click again, while holding the Ctrl-key pressed to unselect. Click the checkbox to check/uncheck the selection. Click the empty tile in the column heading to sort the list. Click again to sort in reverse order.
Property	Displays a list of all properties of the selected entities. Click the *Property* column heading to sort the list; click again to sort in reverse order.
Name	Displays a list of all names of the selected entities. Click the *Name* column heading to sort the list; click again to sort in reverse order.
Category	Displays a list of all categories of the selected entities. Click the *Category* column heading to sort the list; click again to sort in reverse order.
Additional Format	Defines the display format of the item in the export file. Select one or more items, the click the *Additional Format* field of an item to display a dialog box:

	 The layout of the dialog box depends on the selected items. The dialog behaves like the *Format* section of the Field dialog.
Back	Switches back to the Data Extraction - Entities dialog box.
Finish	Extracts the data and saves them to the *.csv file.

Related Commands

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

DbList

Lists information about all entities in the drawing (short for "database listing").

Accessing the Command

command bar: *dblist*

: ***dblist***
Displays entity data in the command bar. Press Esc to quit the command.

Command Options

No options.

Related Commands

List - lists the properties of individual entities in the command bar..
Properties - lists properties of entities in a palette.
Status - reports the status of the drawing.

DdAttE

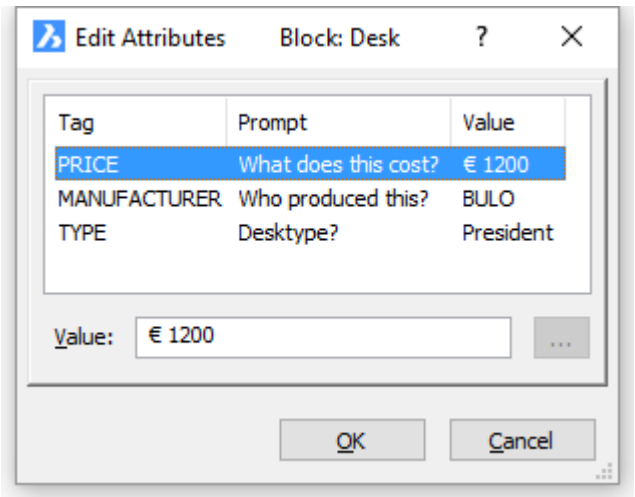
Edits the values of attributes through a dialog box (short for "dynamic dialog attribute editor").

Accessing the Command

command bar: *ddatte*

: ***ddatte***
Prompts you in the command bar:
Select a block: *(Choose a block.)*

Displays a dialog box:



Change attribute value, and then click **OK**.

Command Options

Option	Description
Value	Changes the value of the attribute text.

Related Commands

-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
AttExt - extracts attributes from drawings to data files through a dialog box.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

DdEdit

Edits single-line text, multi-line text, attribute definitions, and attribute text (short for "dynamic dialog editor").

Accessing the Command

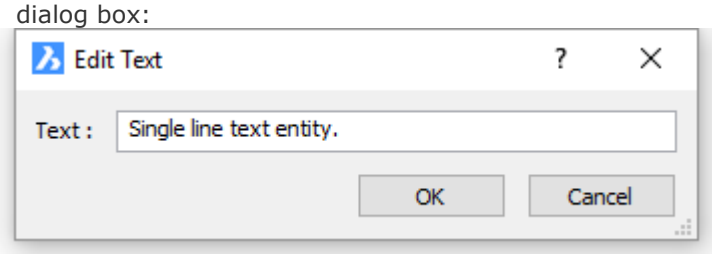
command bar: *ddedit*
alias: *ed*
mouse: double-click: single-line text, multi-line text, attribute definition, or attribute text.

: ***ddedit***

Prompts you in the command bar:

Select entity to modify: (Choose a text entity.)

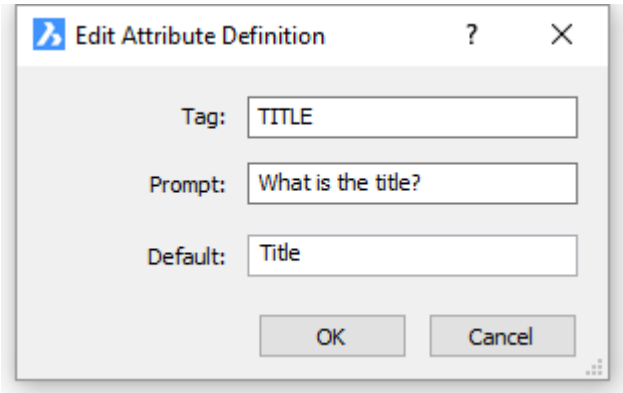
- When you choose single-line text (created by the **Text** command), displays the Edit Text dialog box:



Edit text, and then click OK**.**

Drag the left or right edge of the Edit Text dialog box to widen it. The new width is remembered the next time you open the dialog box.

- When you choose multi-line text (created by the **MText** command), displays the Multiline Text dialog box.
  The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.
- When you choose attribute definition text (created by the **AttDef** command), displays the Edit Attribute Definition dialog box:



Edit attribute definition, and then click OK.

- When you choose attribute text in a block (placed by the **Insert** command), displays the Attribute Editor dialog box; see **EAttEdit** command.

  After exiting the dialog box the value of the TEXTEDITMODE system variable controls whether the command is automatically repeated or not.

- 0: Repeats the command until the Esc-key is hit.
- 1: Does not repeat the command. The command ends when a single text entity is edited.

- 2: Automatic edit mode. Single editing in case of preselected text; multiple otherwise.

Command Options

No options.

Procedures

To edit a text entity
To edit a selection of Mtext entities
To select a different multiline text editor
To edit tag, prompt or default text of an attribute
To edit an attribute definition

Related Commands

Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
DdAtte - edits attribute values.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
Change - changes the position of text.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Properties - modifies the properties of text.
Find - finds and replaces text.

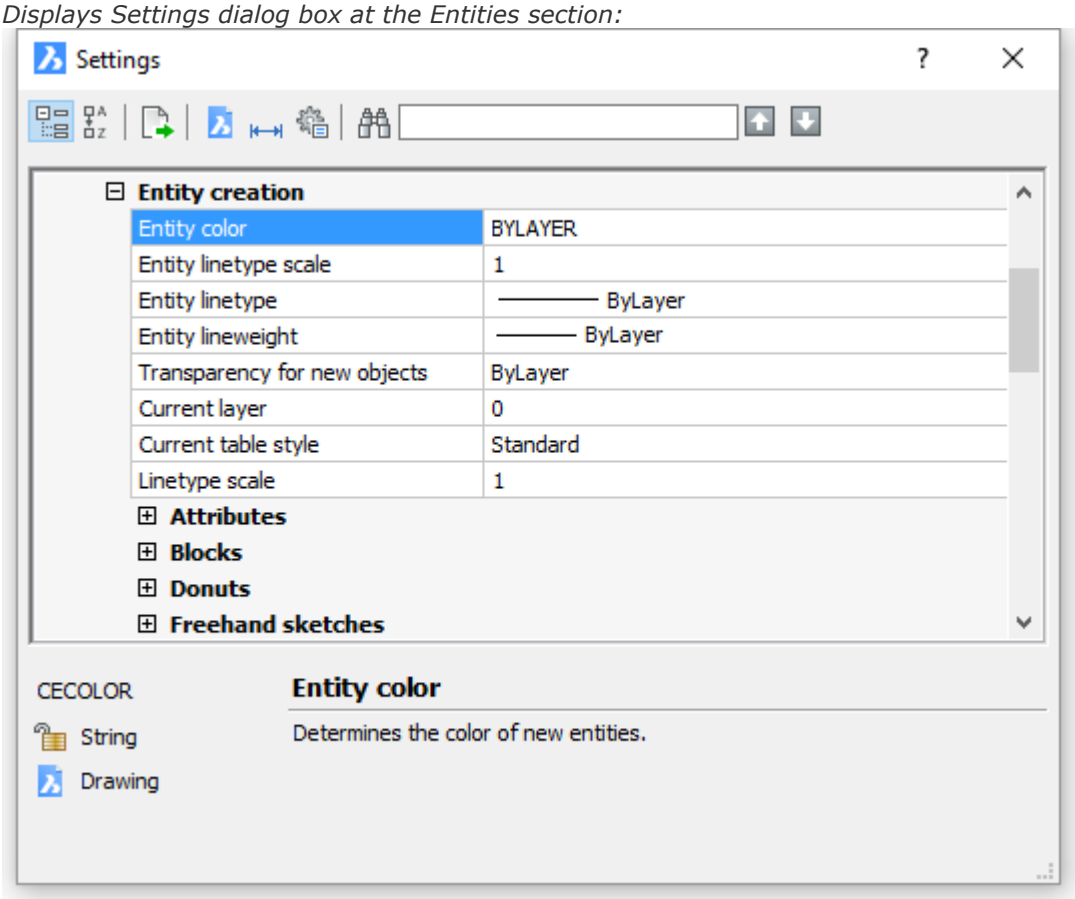
DdEModes

Sets default values for creating entities (short for "dynamic dialog entity modes").

Accessing the Command

command bar: *ddemodes*
alias: *settings*

: ***ddemodes***
Displays Settings dialog box at the Entities section:



*Change settings, and then click **X**.*

Command Options

Option	Description
Entity Color	Specifies the default color of new entities; default is ByLayer, which means that entity colors are governed by layer properties.
Entity Linetype Scale	Specifies the default scale factor for linetypes; default is 1.0000.
Entity Linetype	Specifies the default linetype for new entities; default is ByLayer, which means that entity linetypes are governed by layer properties.
Entity Lineweight	Specifies the default linetype for new entities; default is ByLayer, which means that entity lineweights are governed by layer properties.
Current Layer	Specifies the default layer for new entities; default is layer 0.
Current Table Style	Specifies the default table style for new tables.
Linetype Scale	Specifies the global linetype scale factor; this factor affects entity linetype scale factors.

Related Commands

Color and -Color - set the working color.
Linetype and -Linetype - loads linetype definitions and set the current linetype.
LWeight - sets the current linetype.
Layer and -Layer - create layers and set their properties.
Properties - reports and changes properties through a palette.

DdFilter

Creates a selection set of the entities you select (short for "dynamic dialog filter").

Accessing the Command

command bar: *ddfilter*
alias: *select*

: ***ddfilter***
Prompts you in the command bar:

Select entities to include in set: (Choose one or more entities using any selection mode.)

Select entities to include in set: (Press Enter to exit the command.)

You can access the selection set created by this command by entering "P" (previous) the next time a command prompts you "Select entities."

Command Options

Option	Description
Select entities to include in set	Selects one or more entities using any selection mode.

Related Commands

Properties - filters entities through the Quick Select button.
Select - pre-selects entities based on their location.

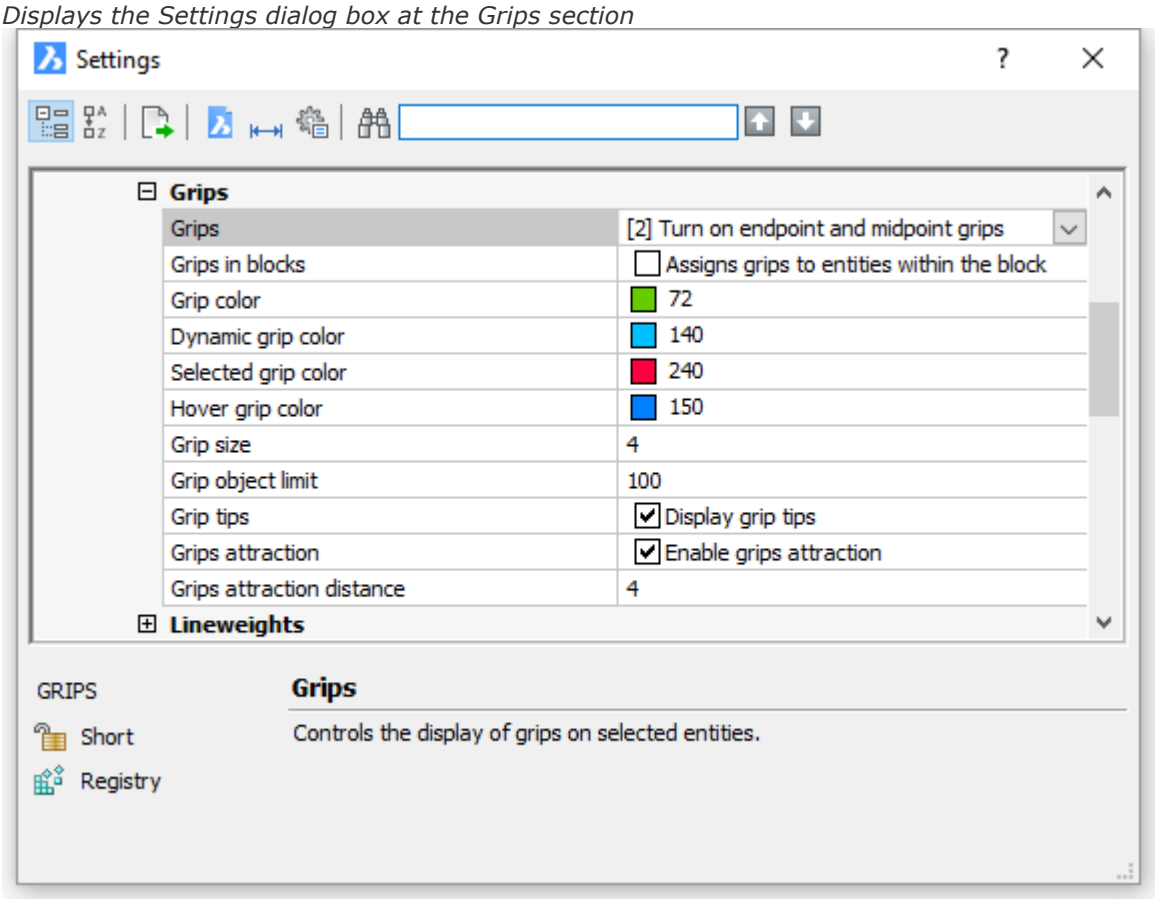
DdGrips

Specifies the properties of grips through the Settings dialog box (short for "dynamic dialog grip settings").

Accessing the Command

command bar: *ddgrips*
transparent: *'ddgrips*
aliases: *gr, settings*

: ***ddgrips***
Displays the Settings dialog box at the Grips section



*Change settings, and then click **X**.*

Command Options

Option	Description
Grips	Sets the display of grips when entities are selected. Allows to control the display of midpoint grips on polyline segments. • 0 - no grips display. • 1 - grips display. On polyline segments, endpoint grips display only. • 2 - grips display. On polyline segments, endpoint and midpoint grips display.
Grips in blocks	Toggles the display of grips inside blocks: • On - display grips on all entities making up the block.

	• Off - display a singe grip at the block's insertion point.
Grip color	Specifies the color of unselected grips; default = index color 72 (green).
Dynamic grip color	Specifies the color of custom grips for dynamic blockx; default = index color 140 (light blue).
Selected grip color	Specifies the color of selected grips; default = index color 240 (red).
Hover grip color	Specifies the color of unselected grips when the cursor pauses over it; default = index color 150 (dark blue).
Grip size	Specifies the size of grips in pixels.
Grip object limit	Specifies the maximum number of entities displayed with grips; default = 100. When the selection set is of more than 100 entities, no grips are displayed. When set to 0, there is no limit to the number of entities.
Grip Tips	Determines whether grip tips display when te cursor hovers over grips on custom entities or dynamic blocks that support grip tips. (not supported yet)
Grips attraction	Toggles the magnetism of entity grips.
Grips attraction distance	Sets the grips attraction distance in pixels.

Related Commands

Select - selects entities.
SelGrips - elects entities and displays grips

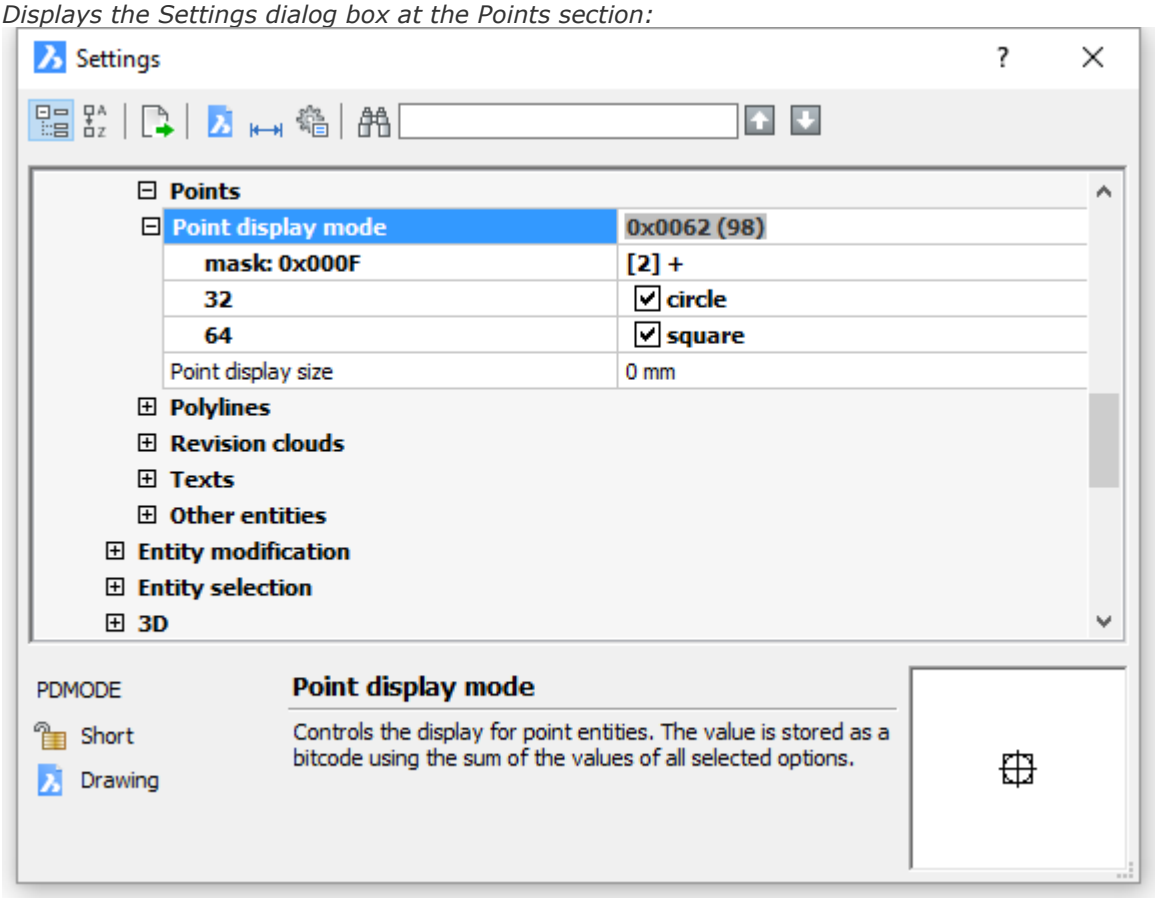
DdPType

Specifies the look and size of point entities, through the Settings dialog box (short for "dynamic dialog point type").

Accessing the Command

command bar: *ddptype*
transparent: *'ddptype*
alias: *settings*

: ***ddptype***
Displays the Settings dialog box at the Points section:



Change settings, and then click X.

Command Options

Option	Description
Point Display Mode	Specifies the look of points: • mask: points are displayed as . (0), + (2), x (3), or ' (4). • Circle - points are displayed as *mask* + circle: • Box - points are displayed as *mask* + square:
Point Display Size	Specifies the size of points: • 0 - points are sized to be 5% of the viewport's height. • Positive number - size of the point in pixels.

	• Negative number - size of the point as a percentage of the viewport's height.

Related Commands

Point - places points in drawings.
Divide - places evenly-spaced points along entities.
Measure - places equi-distant points along entities.

DdSelect

Specifies the properties for selecting entities, through the Settings dialog box (short for "dynamic dialog selection").

Accessing the Command

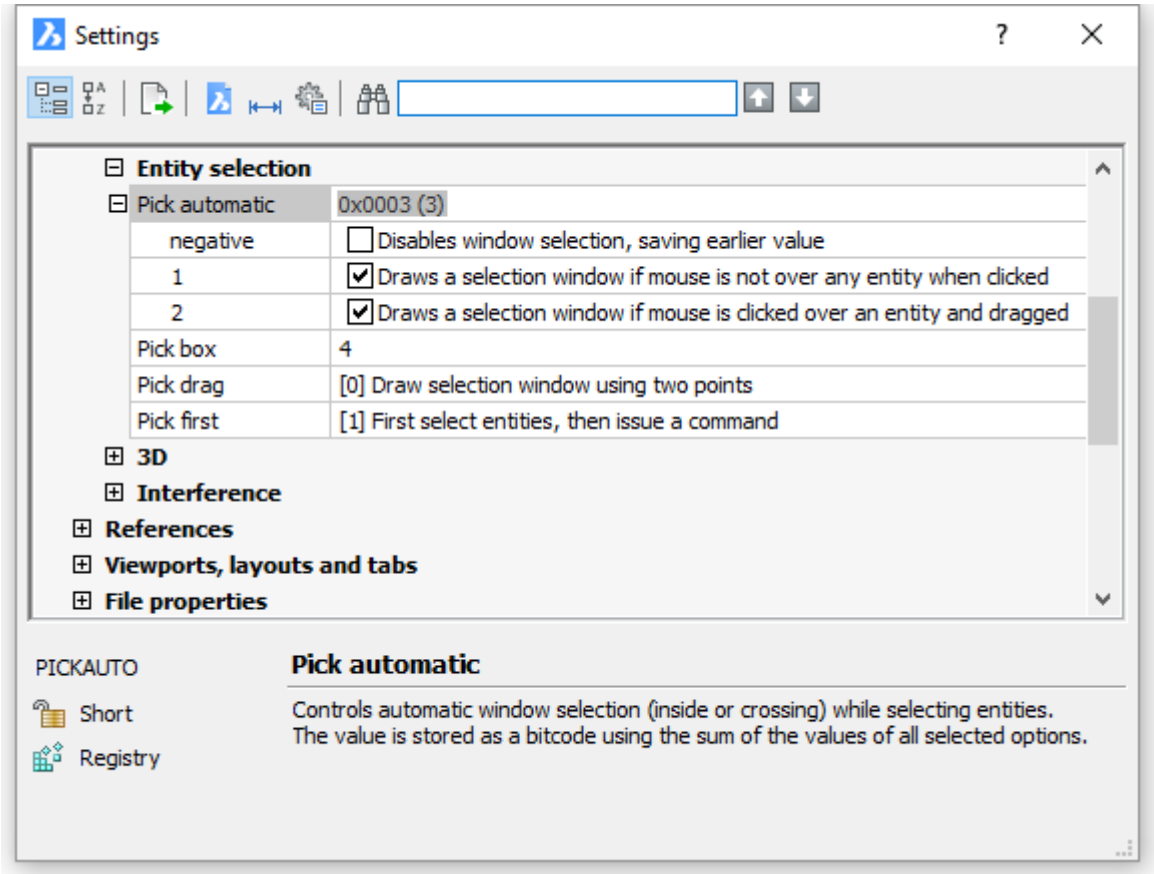
command bar: *ddselect*

transparent: *'ddselect*

alias: *se, settings*

: ***ddselect***

Displays the Entity Selection section of the Settings dialog box:



Change settings, and then click *X*.

Command Options

Option	Description
Pick Automatic	Toggles whether BricsCAD automatically creates window or crossing selections at the 'Select entities' prompt: • negative - Disables window selection when no command is active. Requires a single pick point to select an entity, or the entry of a selection mode option when a command prompts to select entities. • 1 - Enables window selection if the mouse is not over an entity when you click.

	• 2 - Enables window selection if the mouse is clicked when over an entity and dragged. This option is recommended when working in a rendered 3D model.
Pick Box	Specifies the size of the pick box; range is 0 to 50 pixels; default = 3 pixels.
Pick Drag	Determines how selection windows are created: • 0 - Draw selection window using two points; default. • 1 - Draw selection window using dragging.
Pick First	Determines how entities are selected: • 0 - First issue a command, then select entities. • 1 - First select entities, then issue a command; default.

Related Commands

SelGrips - selects entities and displays grips.
Select - selects entities.

DdSetVar

Displays the Settings dialog box (short for "dynamic dialog set variables").

Accessing the Command

command bar: *ddsetvar*
transparent: *'About*
alias: *settings*

: ***ddsetvar***

See the Settings command.

Related Command

SetVar - change the values of system variables through the command bar.
Settings - change the values of system variables through a dialog box.
SettingsSearch - opens the Settings dialog box at the specified category, system variable or user preference.

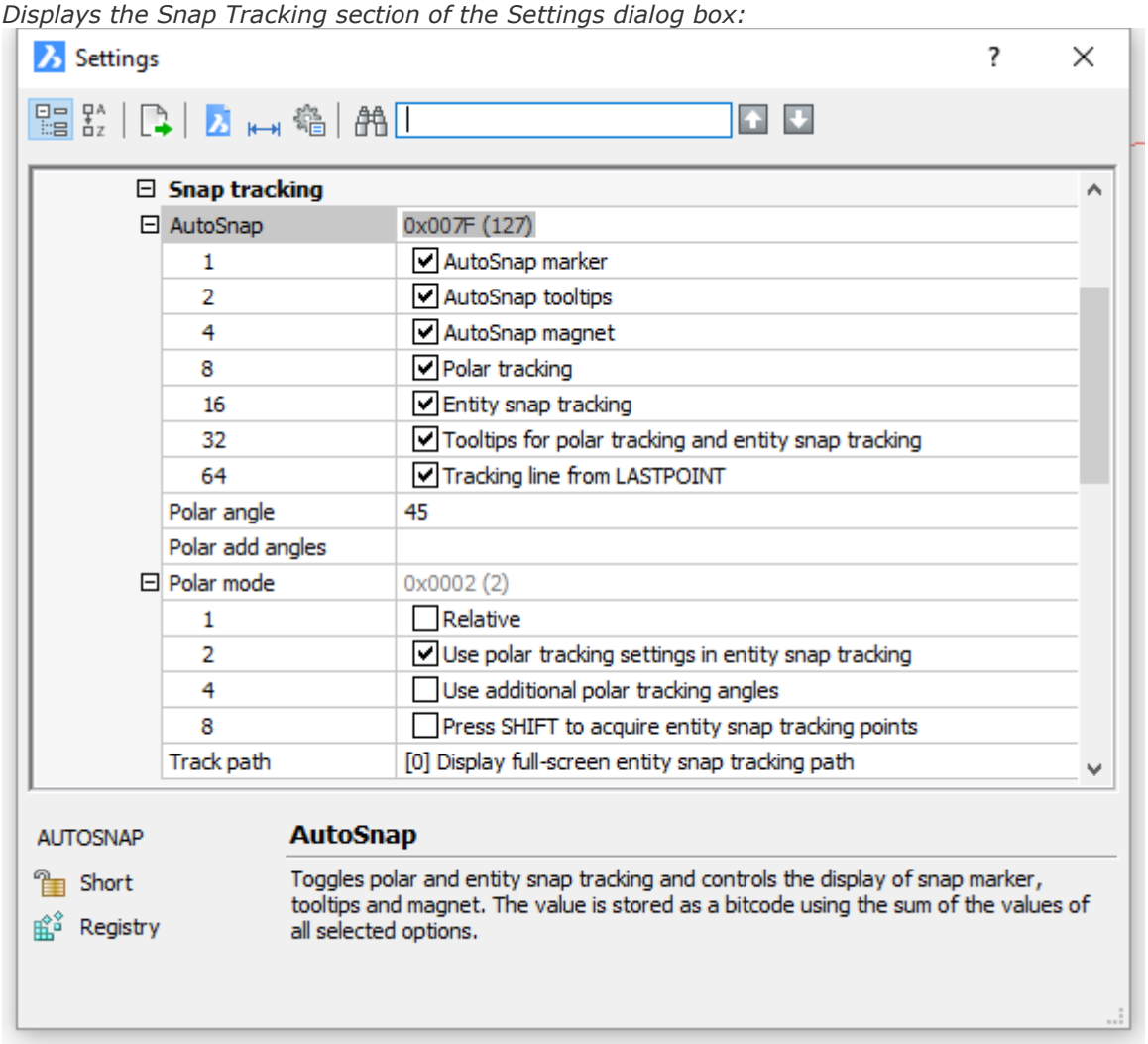
DdSTrack

Sets the properties for snap tracking, through the Settings dialog box (short for "dynamic dialog snap tracking").

Accessing the Command

command bar: *ddstrack*
transparent: *'ddstrack*
alias: *settings*
status bar: *STRACK*

: ***ddstrack***
Displays the Snap Tracking section of the Settings dialog box:



*Change settings, and then click **X**.*

Command Options

Option	Description
AutoSnap	Specifies the look and action of AutoSnap: • AutoSnap Marker - toggles the display of the AutoSnap marker, which indicates the type of entity snap; default = ON. • AutoSnap Tooltips - toggles the display of tooltips adjacent to the entity snap location; default = ON. • AutoSnap Magnet - toggles whether AutoSnap jumps to the nearest entity snap geometry; default = ON. • Polar Tracking - toggles polar tracking, where BricsCAD shows polar angles; default = ON. • Entity Snap Tracking - toggles entity snap tracking; default = ON. • Tooltips for Polar Tracking and Entity Snap Tracking - toggles the display of tooltips; default = on • Tracking line from LASTPOINT - toggle the display of a tracking line from the most recently specified point; default = ON.
Polar Angle	Specifies the default polar snap increment; default = 90.
Polar Add Angles	Specifies additional user-defined angles for polar snap.
Polar Mode	Specifies how polar mode is handled. The value is stored as the sum of the values of the selected options: • 1 - Measure polar angles from selected entities (relative); default. • 2 - Use polar tracking settings in entity snap tracking. • 4 - Use additional polar tracking angles. • 8 - Press the Shift-key to acquire entity snap tracking points.
Track Path	Determines how the tracking path is displayed. The options are: • Display full-screen entity snap tracking path. • Display entity snap path only between the alignment path and the From point to the cursor location; default. • Do not display polar tracking path. • Do not display polar or entity snap tracking paths.

Related Commands

-OSnap - toggles entity snap modes at the command line.
OSnap - sets entity snaps through the Settings dialog box.

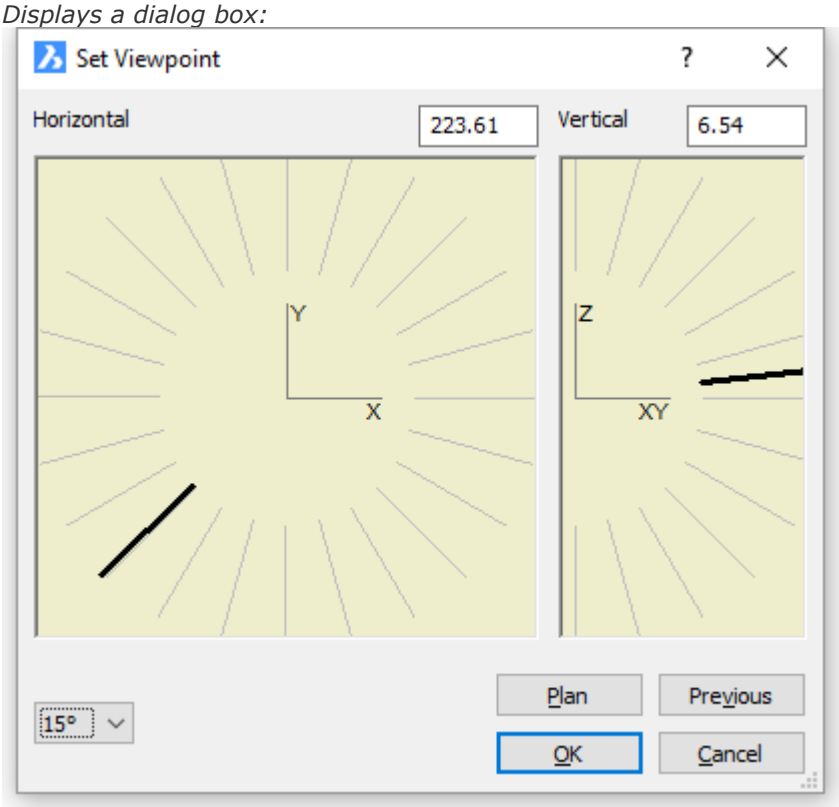
DdVPoint

Sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").

Accessing the Command

command bar: *ddvpoint*
alias: *setvpoint, viewctl, vp*
menu bar: *View | Set Viewpoint*
toolbar: *View |*

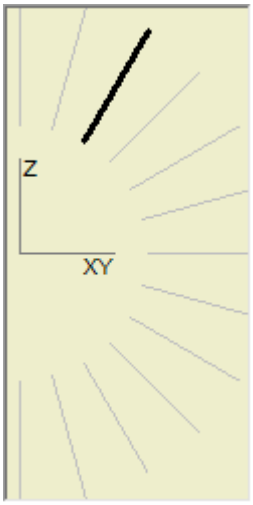
: ***ddvpoint***
Displays a dialog box:



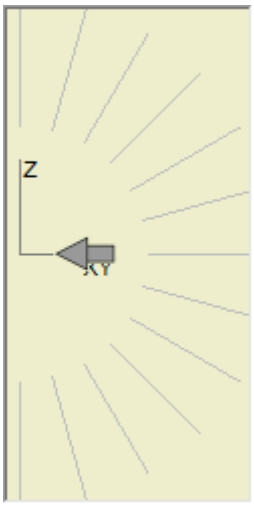
*Choose a viewing angle, and then click **X**.*

Command Options

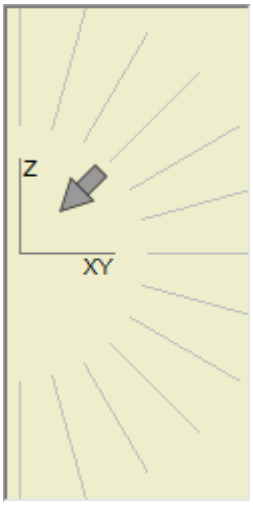
Option	Description
Vertical	Specifies the vertical angle from the XY plane. Click a preset angle or type a value in the *Vertical* field. Vertical 23.00



Click in the inner half circle.

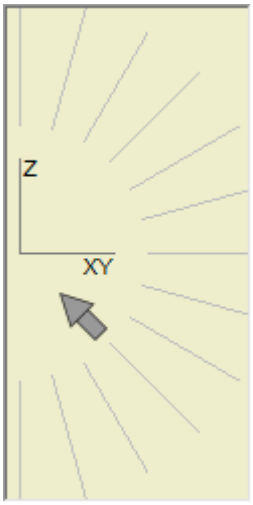


View direction is horizontal.



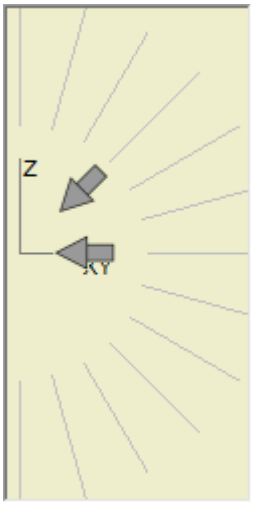
View direction is 45° down.

This option is not available if *Preset Angle*= 45°.



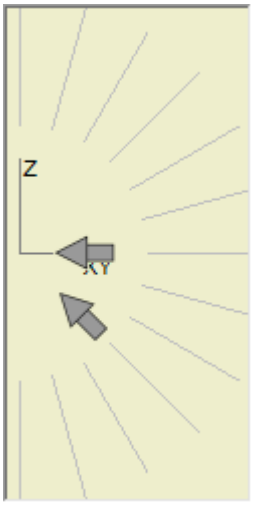
View direction is 45° up.

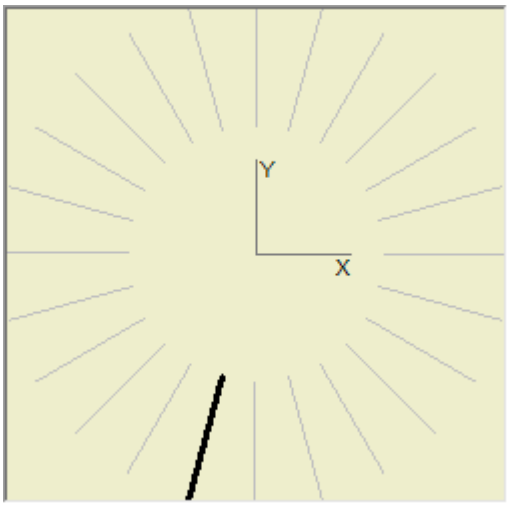
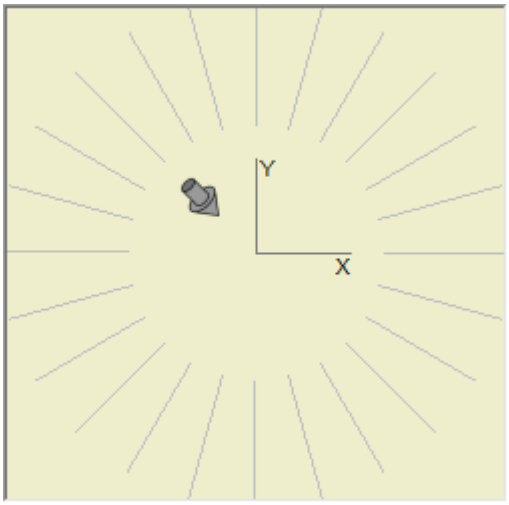
This option is not available if *Preset Angle* = 45°.



View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right.

First select the horizontal arrow, then click the 45° down arrow.



	View direction is 45° up, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. First select the horizontal arrow, then click the 45° up arrow.
Horizontal	Specifies the horizontal angle from the X-axis. Click a preset angle or type a value in the *Horizontal* field. Horizontal 47.00  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 ( ) 
Angle Preset	Sets markers at 5, 15, or 45 degrees; default = 45.
Plan	Reverts the 3D viewpoint to 2D plan view.
Previous	Reverts to the previous viewpoint.
OK	Sets the viewpoint and exits the dialog box.

Look From control

The Look From control allows to select a number of preset views such as orthographic views and isometric views.

Related Commands

VPoint - changes the 3D viewpoint of the drawing through the command bar.
DView - changes the 3D viewpoint interactively, and turns on perspective mode.
RtRot - rotates 3D drawings in real-time

Delay

Delays execution of the next command; meant for use with scripts only.

Accessing the Command

command bar: *delay*
transparent: *'delay*

: ***delay***
Prompts you in the command bar:
Milliseconds to delay: *(Enter a number between 0 and 2,147,483,627 -- approximately 24 days.)*

Command Options

Option	Description
Milliseconds to Delay	Specifies the amount of time BricsCAD waits before continuing to the next command in the script; specified in milliseconds (1/1000th of a second).

Related Commands

RecScript - records scripts to SCR files.
StopScript - stops recording the script, and saves it to an SCR file.
Script - loads and runs a SCR script file.
RScript - reruns the script.
Resume - resumes a stopped script.

DelConstraint

Removes all dimensional and geometrical constraints from an entity.

Accessing the Command

command bar: *delconstraint*
menu bar: *Parametric | Delete 2D Constraints*
toolbar: *2D Constraints |* ✕

: ***delconstraint***
Prompts you in the command bar:

Select entities to delete all constraints: (select an entity or entities)
Entities in set: 1

Select entities to delete all constraints: (select more entities or press Enter to delete all constraints from the selection set)

The dimensional and geometric constraints of the selected entities are deleted.

Command Options

No options

Related Commands

-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.

DelEData

Deletes extended entity data from the selected entity (short for "delete entity data").

Accessing the Command

command bar: *deledata*
transparent: 'deledata
menu bar: *Tools | Entity Data | Delete Entity Data*
toolbar: *Entity Data |*

: ***d*eledata**
Prompts you in the command bar:

Delete Entity Data: Name of application or [?]: (Enter application name, or type ?.)
Select entities to delete Entity Data: *(Choose one or more entities.)*
Entities in set: 1

Select entities to delete Entity Data: (Select more entities or press Enter to end command.)

Command Options

Option	Description
?	Lists names of applications loaded in this session of BricsCAD.
Name of Application	Specifies the name of the application associated with the extended entity data.
Select Entities to Delete Entity Data	Specifies the entities from which to erase extended entity data.

Related Commands

CopyEData - copies extended entity data to other entities.
EditEData - creates and edits extended entity data.
MoveEData - moves extended entity data to other entities.
ReAssocApp - associates applications with entities.

DesignTable

Creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser and parametric component properties.

Accessing the Command

command bar: *designtable*
menu: *Parametric | Design Table from file* and *Empty design Table*
toolbars:
Parameters |
Design Table | and
ribbon panel: *3D Constraints |* (Sheet Metal Workspace: *Sheet Metal* and *Assembly* tabs - Mechanical workspace: *Solid*, *Surface* and *Assembly* tabs)

: ***designtable***
Prompts you in the command bar:
Create design table [From file/Empty] <From file>: *(Choose an option.)*

Command Options

Option	Description
From file	Creates a design table by importing a .csv file (Comma Separated Values file). Please make sure the list separator character on your system is the same as the one used in the CSV file. Displays the *Select design table file* dialog box. Select the .csv file and click the *Open* button or double click the file name.

Empty	Creates an empty design table; prompts you:
	Enter design table name <DesignTable_1> - type a name or press Enter to accept the default name.
	Enter parameter names ('*' to link all constant parameters) <*> -
	Do one of the following:
	• Press Enter to add all parameters that have a constant value (= do not depend on another parameter).
	• Enter the names of the parameters to include in the design table.
	  • Press Enter after each name.
	  • Press Enter twice to stop adding names and create the design table.

Procedures

Creating a Design Table from a .csv file
Direct Lookup in a Design Table
Reverse Lookup in a Design Table
Adding a New Configuration to a Design Table
Creating an Empty Design Table
Using a Parametric Component with Design Tables

Related Commands

-DesignTableEdit - allows to edit designtables.

-BmParameters - allows to list and edit the parameters of inserted components and associative array
-Parameters - manages dimensional constraint parameters in the current drawing.
MechanicalBrowserOpen - opens the Mechanical Browser panel.

Dish

This command is obsolete. It still exists for backward compatibility only.
Please use ai_dish instead.

Dist

Reports the distance and angle between two points.

Accessing the Command

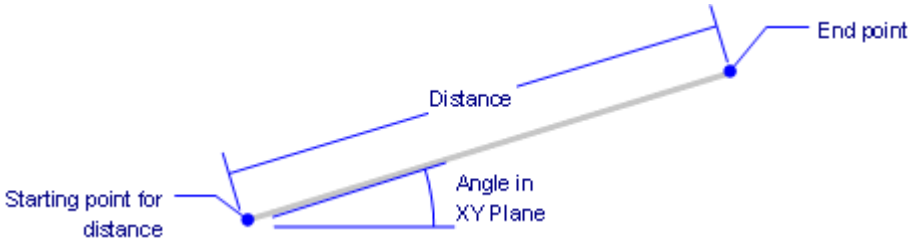
command bar: *dist*
transparent: *'dist*
alias: *di*
menu bar: *Tools | Inquiry | Distance*
toolbar: *Inquiry |*

: ***dist***
Prompts you in the command bar:

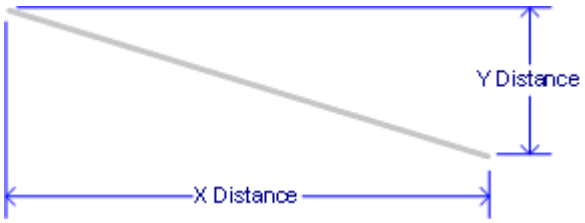
Starting point for distance: (Pick a point, or enter coordinates.)

End point: (Pick another point, or enter coordinates.)
Sample response:

Distance = 4.3167, Angle in XY Plane = 220, Angle from XY Plane = 0



Delta X = -3.3302, Delta Y = -2.7466, Delta Z = 0.0000



NOTE If DRAGSNAP = ON and Dynamic Dimensions are enabled the distance displays in the dynamic entry field when hovering over the second point using an appropriate entity snap.

Command Options

Option	Description
Starting Point for Distance	Specifies the first point from which the measurement takes place.
End Point	Specifies the other point.

Related Commands

Area - reports the area, circumference, and perimeter of entities.
Id - reports the x,y,z coordinates of picked points.
List - lists the area and other parameters of entities in the command bar.
MassProp - reports the properties of solid models and regions.
Properties - lists the area and other parameters of entities in a palette.

Distantlight

Creates a distant light.

Accessing the Command

command bar: *distantlight*
menu bar: *View | Rendering | Lights | New distant light*
toolbar: *Lights |*

: ***distantlight***
Prompts you in the command bar:

Specify light direction or [Vector] <0,0,0>: (Type V then press Enter to choose Vector, specify a point or press Enter to accept the light direction from 0,0,0.)

Specify vector direction <0.0000,-0.0100,1.0000>: (Type the vector direction at the command prompt and press Enter to confirm.)

Specify light direction TO <1,1,1>: (Specify a point or press Enter to accept point 1,1,1)

If the LIGHTINGUNITS system variable is 0 (No lighting units), the following prompt displays:

Enter an option to change [Name/Intensity/Status/shadoW/Color/eXit] <eXit>: (Enter an option or press Enter to exit.)

If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), the following prompt displays:

Enter an option to change [Name/Intensity factor/Status/Photometry/shadoW/filterColor/eXit] <eXit>: (Enter an option or press Enter to exit.)

Command Options

Option	Description
Name	Enter light name <Distantlight1>: (Enter a name or press Enter to accept the default name.)
Intensity Factor Intensity	Enter intensity (0.00 - max float) <1.0>: (Enter the intensity. Values between 0.00 and the maximum value supported by your system are accep²²ted.)
Status	Enter status [ON/OFf] <ON>: (Choose ON to include the light when calculating a rendering.)

Photometry	**Enter a photometric option to change [Intensity/Color/eXit] <Intensity>:**
	• Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>: (Enter the intensity expressed in Candela (Cd).)
	  • Enter Flux (Lm) : (Enter the luminous flux expressed in Lumen (Lm).)
	  • Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>: (Enter the illuminance expressed in Lux (Lx) or Foot-candle (Fc).)
	  • Enter Distance <1.0>: (Enter a distance expressed in drawing units.)
	• Enter color name or enter an option [?/Kelvin] <D65>:

- Enter color name(s) to list <*>: (Enter a color name from the list. Type * to see the list)
  - D65
  - Fluorescent
  - Coolwhite
  - Whitefluorescent
  - daylightfluorescenT
  - Incandescent
  - Xenon
  - Halogen
  - Quartz
  - Metalhalide
  - mErcury
  - Phosphormercury
  - highpressureSodium
  - Lowpressuresodium
- Enter Kelvin temperature <3600.0>: (Enter a color temperature expressed in degrees Kelvin (K).)

Candela: The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela.

Lumen: The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr

As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens.

Illuminance: In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human brightness perception.

If LIGHTINGUNITS=2 (International lighting units):
In SI derived units illuminance is measured in lux (lx) or lumens per square metre.

If LIGHTINGUNITS=1 (American lighting units):
The foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux,

	although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux. Kelvin temperature: Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K). Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700&endash;3,000 K) are called warm colors (yellowish white through red).

Shadow	Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled]: <Sharp>: (Enter a shadow setting.)
	• Off: Disables the calculation of shadows for the light.
	• Sharp: Displays shadows with sharp edges. Use this option to increase performance.
	• Soft mapped: Displays realistic shadows with soft edges.
	Prompts you: Enter map size [64/128/256/512/1024/2048/4096] <256>:
	• Soft sampled: Displays realistic shadows with softer shadows based on extended light sources.
	Prompts you: Enter an option to change Shape/sAmples/Visible/<eXit>:
	  • Enter shape [Disk/Rect] <Disk>:
	  • Enter radius <0.0>: (Enter the radius of the disk.)
	  • Enter Length <0.0>: (Enter length of the rectangle.)
	  • Enter Width <0.0>: (Enter width of the rectangle.)
	  • Enter Shadow Sample <16>:
	  • Enter Shape Visibility [Yes/No] <No>:

Filter Color
Color
	Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>: Defines the color of the light.
	  • True Color (RGB): The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values).
	  • Index color: Specifies an AutoCAD Color Index color.
	  • HSL: Specifies the color using three parameters Hue, Saturation and Lightness.

Exit	Concludes the command and creates the light.

Related Commands

Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Pointlight - creates a point light.
Render - calculates a photorealistic image of the 3D model.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.

Weblight - creates a web light.

Divide

Places points or blocks along entities.

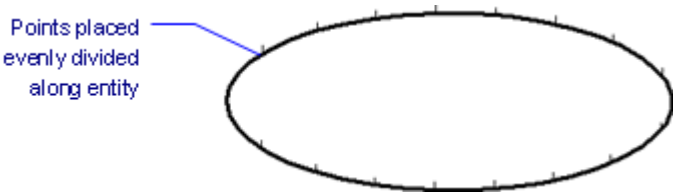
Accessing the Command

command bar: *divide*
alias: *div*
menu bar: *Modify | Divide*
toolbar: *Modify |*

: ***divide***
Prompts you in the command bar:

Select entity to divide: (Choose a line, polyline, spline, arc, circle, or ellipse.)

Number of segments [insert Block]: (Enter a number, or type B.)



Command Options

Option	Description
Select Entity to Divide	Chooses a single entity: a line, polyline, spline, arc, circle, or ellipse.
Number of segments	Specifies the number of segments. Prompts you: Number of segments - type an integer. Use the PDMODE and PDSIZE system variables to set the display of the dividing points.
Blocks	Uses blocks instead of points. Prompts you: Name of block to insert [? to list] - Type the name of the block or press ? to list the available blocks in the Prompt History window. **Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-Align blocks>:** • Yes - rotates blocks to match the entity. • No - does not rotate blocks, which maintain their orientation. Number of segments - type an integer.

Procedures

To divide an entity using points
To divide an entity using blocks

Related Commands

Measure - places equi-distant points along entities.
Point - places points in drawings.
DdPType - specifies the look and size of point entities through the Settings dialog box

Dome

This command is obsolete. It still exists for backward compatibility only.
Please use ai_dome instead.

Donut

Draws circles with width.

Accessing the Command

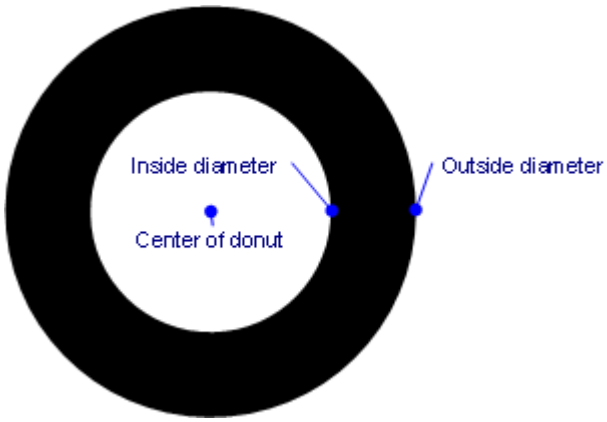
command bar: *donut*
alias: *do, doughnut*
menu bar: *Draw | Donut*
toolbar: *Draw |*

: ***donut***
Prompts you in the command bar:
Inside diameter of donut [2 Point/3 Point/Tangent Tangent Radius] <0.5>: *(Pick a point, enter a value, or enter an option.)*

Outside diameter of donut <1>: *(Specify a value, or pick two points.)*

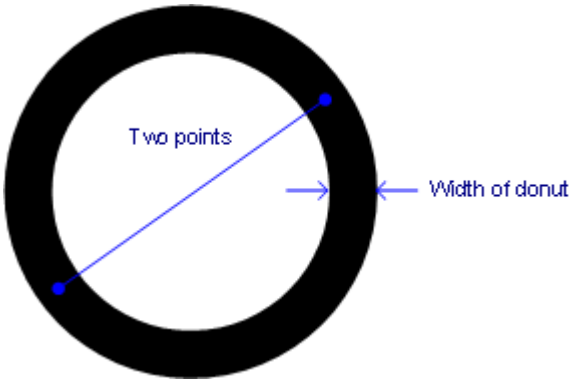
Center of donut: (Specify coordinates, or pick a point in the drawing.)

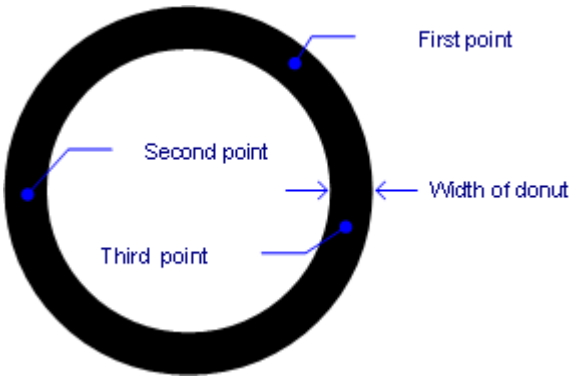
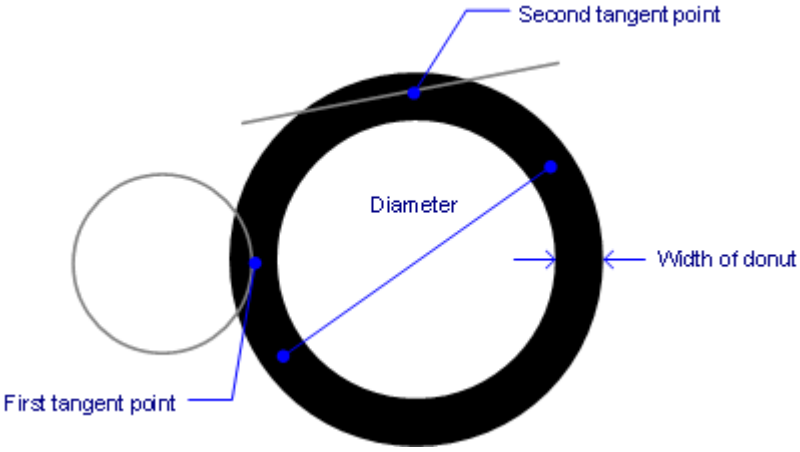
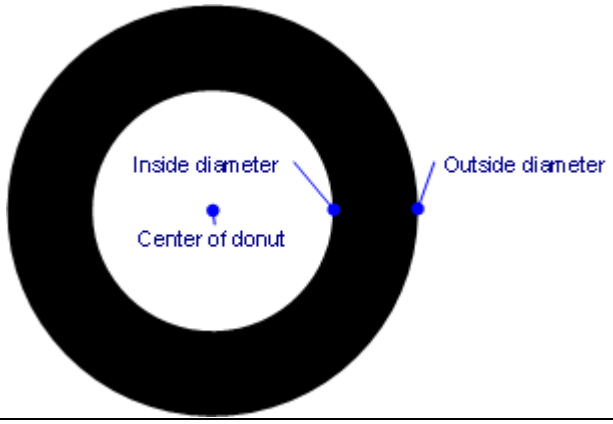
Center of donut: (Specify coordinates, or pick a point in the drawing, or press Enter to conclude the command.)



Donuts are made of two wide polyline arcs.

Command Options

Option	Description
2Point 	Defines the donut by two points on its circumference; prompts you: Width of donut - enter a value, or pick two points. First point on diameter - enter coordinates, or pick a point. Second point on diameter - enter coordinates, or pick another point. The abbreviation for this option is 2P.

3Point  	Defines the donut by three points on its circumference; prompts you: Width of donut - enter a value, or pick two points. First point on donut - enter coordinates, or pick a point. Second point - enter coordinates, or pick another point. Third point - enter coordinates, or pick another point. The abbreviation for this option is 3P.
Tangent Tangent Radius  	Defines the circle by two points tangent to other entities and the radius; prompts you: Width of donut: (Enter a value, or pick two points. Diameter of donut - enter a value, or pick two points. Select first tangent point - enter coordinates, or pick an entity. Select second tangent point - enter coordinates, or pick another entity. The abbreviation for this option is R.
Diameter of donut  	Defines the donut by inner and outer diameter, and a center point; prompts you: Outside diameter of donut - specify a value, or pick two points. Center of donut - specify coordinates, or pick a point in the drawing.

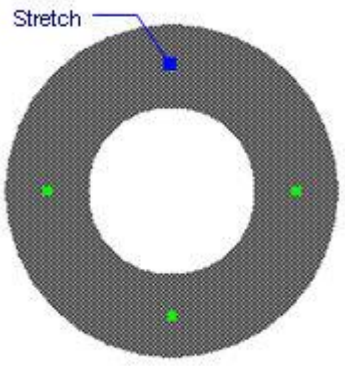
Procedures

Drawing donuts

Grips Editing

Donuts can be edited directly through grips:

1. Select the donut. Notice that it has four grips.
2. Drag a grip to stretch the donut (increase or decrease its diameter).



Related Commands

Circle - draws regular circles.
PLine - draws polyline segments and arcs.
Rectang - draws rectangles out of polylines.
PEdit - edits polylines.
3dPoly - draws polylines with x,y,z coordinates.
Polygon - draws regular polygons out of polylines.

DragMode

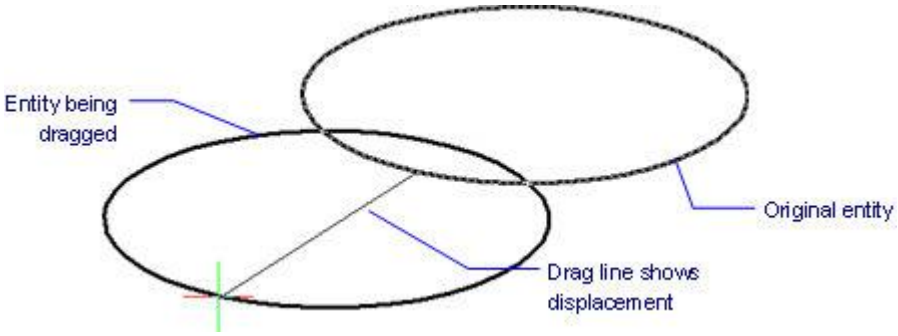
Controls the appearance of objects while being dragged.

NOTE This command was useful in the early days of CAD when computers were slow and had difficulty updating the screen quickly enough to show objects being dragged. This command is no longer necessary, and has been kept only for compatibility.

Accessing the Command

command bar: *dragmode*
transparent: *'dragmode*

: ***dragmode***
Prompts you in the command bar:
[ON/OFf/Auto] <Auto>: *(Enter an option.)*



Command Options

Option	Description
ON	Turns on dragmode; same as Auto mode.
OFF	Turns off dragmode; dragged objects and the dragline are never shown on the screen.
Auto	Turns on drag mode; dragged objects are always shown.

Related Command

Settings - sets drag mode through the Drawing Display section of the Settings dialog box.

DrawOrder

Changes the display order of overlapping entities.

Accessing the Command

command bar: *draworder*
alias: *dr*
menu bar: *Tools | Draw Order*
toolbar: *Draw Order |*

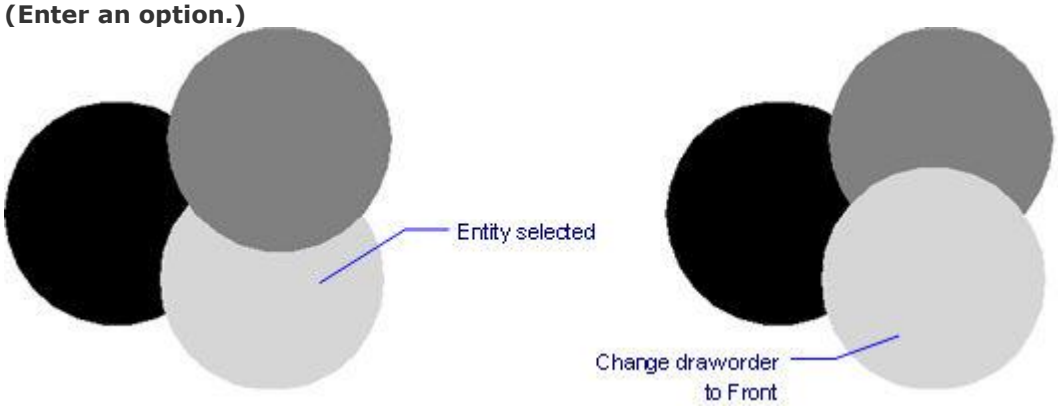
: ***draworder***
Prompts you in the command bar:

Select entities to change draw order: **(Select one or more entities.)**

Entities in set: 1

Select entities to change draw order: *(Select one or more entities, or press Enter.)*

Change draw order [Above/Under/Clear all orders/bring to Front/send to Back] <Send to back>: **(Enter an option.)**



Command Options

Option	Description
Select Entities to Change Draw Order	Selects one or more entities; this command is meant to work with overlapping objects.
Above	Moves the entities above other entities, but not necessarily to the top (front) of display order.
Under	Moves the entities below other entities, but not necessarily to the bottom (back) of display order.
Clear all orders	Clears assigned display orders; entities are displayed as they were originally created. The last created entity is displayed on top of one created earlier.
Bring to Front	Moves the entities' display order above all other entities.
Send to Back	Moves the entities' display order below all other entities.

Procedures

Adjusting the Draw Order

Related Commands

DrawOrderByLayer - sets draw order of overlapping objects through layer names.
HatchToBack - sets the draw order of all hatch entities in the drawing to display behind all other entities.
TextToFront - sets the draw order of all texts and dimensions in the drawing to display in front of all other entities.
BHatch and Hatch - set the draw order of hatch patterns relative to other entities through a dialog box.
-BHatch and -Hatch - set the draw order of hatch patterns relative to other entities through the command bar.

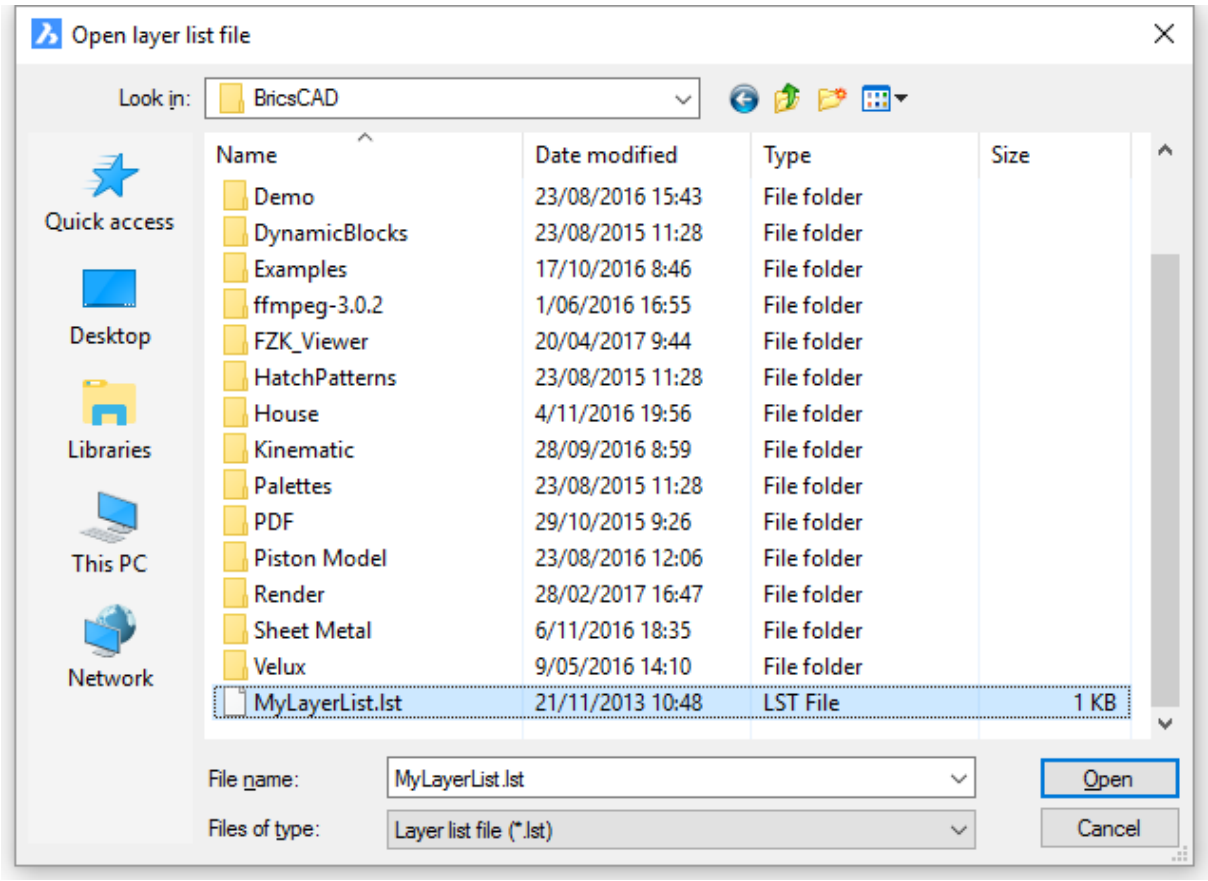
DrawOrderByLayer

Controls the draw order of overlapping objects through layer names.

Accessing the Command

command bar: *draworderbylayer*

: ***draworderbylayer***
Displays a dialog box:



Choose a *.lst layer list file, and then click Open.

The file must be created by the user. The *.lst file is an ASCII text file that contains a list of layer names in the following format:

LayerName1

LayerName2

...

LayerNameN

The DrawOrderByLayer command draws the layers in the order defined in the *.lst file: first the entities on layer LayerName1, then the entities on LayerName2, etc. The entities on LayerNameN end up on top.

Wildcard characters are supported in layer names. All entities on layers that match the selection criterium are then drawn. As a result a filter that is preceded by a more general filter will do nothing. E.g. filter at* after filter a* has no effect because all entities on layers starting with 'a' are

already drawn. But filter a* after *at, will first draw the entities on layers starting with 'at', then all entities on the leftover layers starting with 'a'.

Command Options

Option	Description
Open	Opens the *.lst file.

Related Commands

DrawOrder - sets draw order of overlapping objects through the command bar.
BHatch and Hatch - set the draw order of hatch patterns relative to other entities through a dialog box.
-BHatch and -Hatch - set the draw order of hatch patterns relative to other entities through the command bar.

DSettings

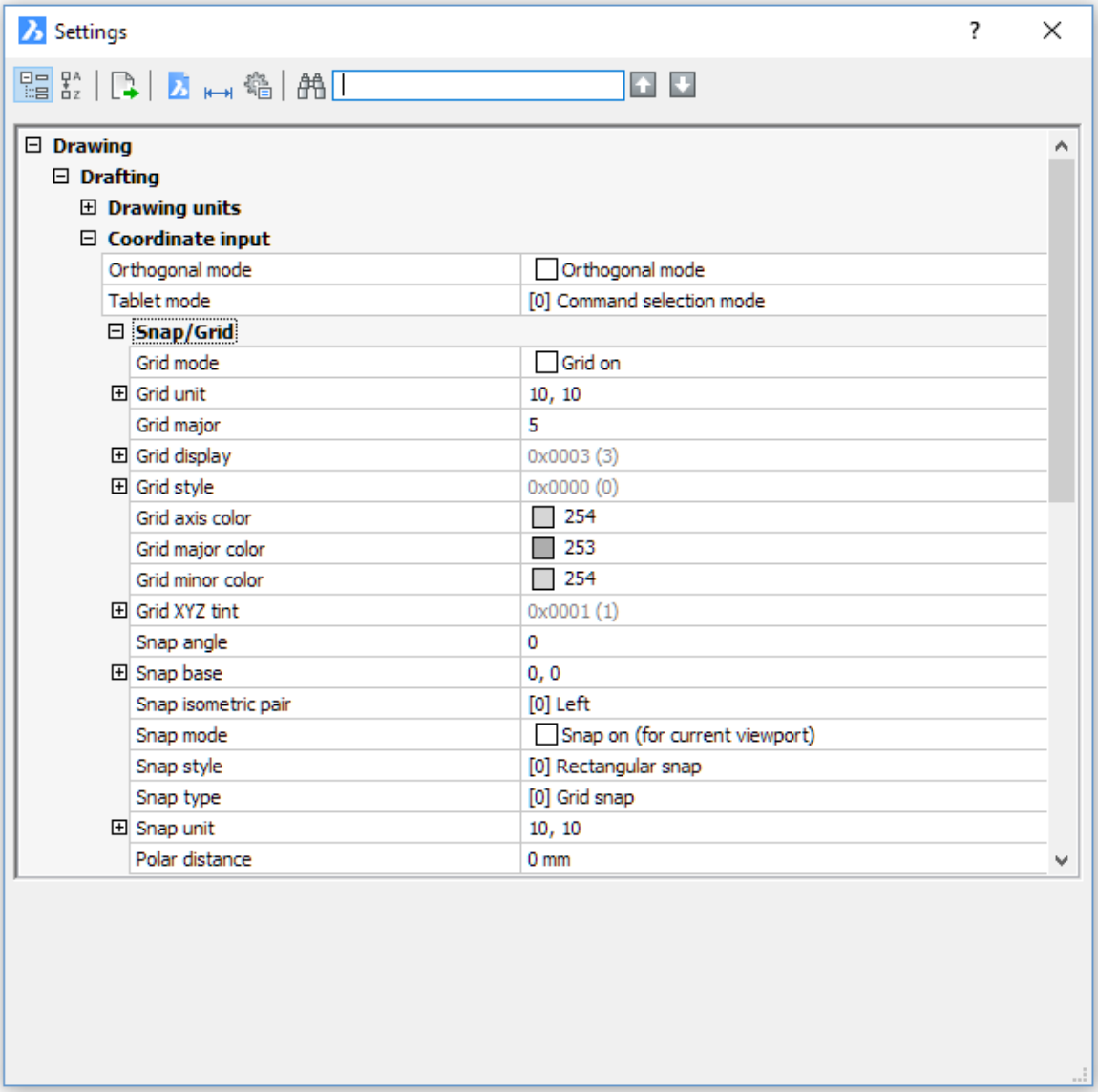
Displays the Settings dialog box for drafting settings (short for "drafting settings").

Accessing the Command

command bar: *dsettings*
transparent: *'dsettings*
alias: *settings*

: ***dsettings***

D*isplays the Settings dialog box at the Drafting section:*



Change settings, and then click X.

Command Options

Coordinate Input	Description
Orthogonal Mode	Toggles orthogonal mode: • On - constrains cursor movement to the vertical and horizontal. • Off - moves cursor freely. Hold down the Shift key to turn on ortho mode temporarily; alternatively, double-click ORTHO on the status bar.
Tablet Mode	Toggles tablet mode: • Digitizing mode - uses tablet for drawing input. • Command selection mode - uses tablet for command input.
Snap/Grid	
Grid Mode	Toggles the grid display: • On - displays the grid. • Off - does not display the grid. To toggle the grid display, you can click GRID on the status bar.
Grid Unit	Specifies the spacing between grid dots: • X - specifies the distance in the x direction. • Y - specifies the distance in the y direction.
Grid Major	Specifies the frequency of manor gridlines versus minor gridlines.
Grid Display	Specifies how the grid is displayed. • Display beyond limits area: If checked, the grid displays on the entire drawing. • Adaptive grid display: If checked, the number of grid lines is adapted to the zoom. Otherwise the grid does not display if the grid is too dense. A warning displays in the command bar: "The grid spacing is too small to display in this view". • Allow subdivision below grid spacing: If checked, extra grid lines display when zooming in closely. • Follow dynamic UCS: If checked, the grid displays in the XY-plane of the dynamic UCS (not yet implemented).
Grid Style	Specifies whether the grid is displayed with dots or lines in Model Space, Block Editor and/or Layouts.
Grid Axis Color	Sets the color of the grid lines on the X-axis and Y-axis of the current coordinate system; default = index color 251.
Grid Major Color	Sets the color of the major grid lines; default = index color 252.
Grid Minor Color	Sets the color of the minor grid lines: default = index color 253.
Grid XYZ tint	Specifies whether the grid axis lines, the major grid lines and/or the minor grid lines display in the color of the X-axis (COLORX) and the Y-axis (COLORY).
Snap Angle	Rotates the snap and grid relative to the current UCS.
Snap Base	Locates the origin of the snap and grid relative to the current UCS: • X - specifies the x coordinate.

	• Y - specifies the y coordinate.
Snap Isometric Pair	Specifies the current isometric plane: • Left. • Top. • Right.
Snap Mode	Toggles snap mode: • On - constrains cursor movement to the snap distance. • Off - allows cursor to move freely. To toggle snap mode, click SNAP on the status bar.
Snap Style	Specifies the snap style: • Rectangular snap - sets snap and grid to rectangular. • Isometric snap - sets snap and grid to isometric; adds the Isocircle option to the Ellipse command.
Snap Type	Specifies the snap type: • Grid snap - restricts cursor movement to x and y snap distances. • Polar snap - restricts cursor movement to angles.
Snap Unit	Specifies the spacing between snap points: • X - specifies the distance in the x direction. • Y - specifies the distance in the y direction.
Polar Distance	Specifies the polar distance; effective only when Snap Type is set to Polar Snap.

Related Commands

DdSetVar - changes the values of system variables through a dialog box.
SetVar - changes system variables through the command bar.

DView

Changes the 3D viewpoint interactively, and turns on perspective mode (short for "dynamic view").

Accessing the Command

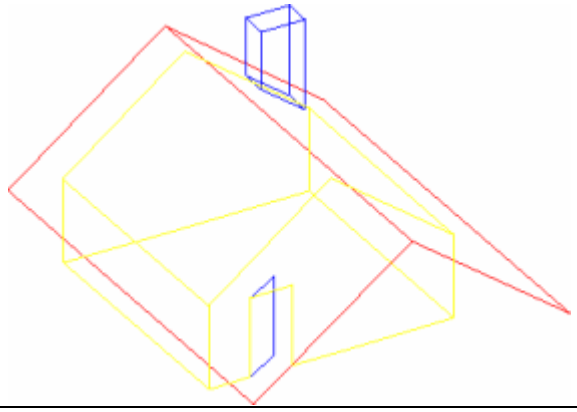
command bar: *dview*
transparent: not available
alias: *dv*
menu bar: *View | Define View*
toolbar: *View |*

: ***dview***
Prompts you in the command bar:

Select entities or <use DVIEWBLOCK>: (Press Enter to display the DVIEWBLOCK, or select entities.)

DView: CAmera/TArget/TWist/Distance/POints/CLip/Off/Hide/PAn/Zoom/Undo/<eXit>: (Enter an option, or press Enter to exit the command.)

Command Options

Option	Description
ENTER to display *DVIEWBLOCK*	Displays a built-in 3D block. 
Select Entities To See In Preview	Chooses selected entities to be previewed by this command.
CAmera	Specifies the camera (eye) location; prompts you: • **Specify camera location, or enter angle from XY plane, or [Toggle (angle in)] <current angle>** Move the mouse to specify the camera location, click to confirm; or key in the angle from the XY plane; Press ***T + Enter*** to toggle to ***angle in XY plane***. • **Specify camera location, or enter angle in XY plane from X axis, or [Toggle (angle from)] <current angle>** Move the mouse to specify the camera location, click to confirm; or key in the angle in the XY plane. Press ***T + Enter*** to toggle to ***angle from XY plane***.
TArget	Specifies the target (look at) location; prompts you: • **Specify camera location, or enter angle from XY plane, or [Toggle (angle in)] <current angle>** Move the mouse to specify the camera location, click to confirm;

	or key in the angle from the XY plane; Press ***T + Enter*** to toggle to ***angle in XY plane***. • **Specify camera location, or enter angle in XY plane from X axis, or [Toggle (angle from)] <current angle>** Move the mouse to specify the camera location, click to confirm; or key in the angle in the XY plane. Press ***T + Enter*** to toggle to ***angle from XY plane***.
TWist	Sets the twist (view rotation about Z axis) angle; prompts you: • **New view twist <current angle>** - Move the mouse to specify the camera location, click to confirm; or specify the angle.
Distance	Specifies the camera to target distance; prompts you: • **New camera-to-target distance <current distance>** Move the mouse to specify the camera to target distance, click to confirm; or key in the distance. • The *Distance* option switches perspective mode on.
POints	Specifies the camera and target location; prompts you: • **Enter target point** - specify the point using the mouse, or enter x,y,z coordinates. • **Enter camera point** - specify the point using the mouse, or enter x,y,z coordinates.
CLip	Sets the clipping distances, which cut off the front and back parts of the model; prompts you: • **Enter clipping option: Front/Back/<Off>** - type F, B, or O.
Off	Turns off perspective mode. Use the *Distance* option to turn perspective mode on. The *Perspective* system variable toggles the perspective property of a view.
Hide	Removes hidden lines from the model.
PAn	Interactively pans around the drawing; prompts you: • **Displacement base point -** specify the point, or enter x,y,z coordinates. • **Second point** specify the point, or enter x,y,z coordinates.
Zoom	Zooms in and out interactively; prompts you: • **View size -** enter a number.
Undo	Undoes the last view change.
eXit	Exits the command.

Related Commands

Camera - creates perspective views.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
RtRot - rotates 3D drawings in real-time

Dwfout

Saves the drawing in DWF and other formats (short for "drawing Web format output").
DwfOut is an alias for the Export command

Accessing the Command

command bar: *dwfout*
menu bar: *File | Export*
alias: *Export*

: ***dwfout***

See the Export command.

DWF files can be viewed with the DesignReview software, available free from www.autodesk.com/designreview.

Related Commands

Export - exports drawings in a variety of formats.
DxfOut - exports drawings in DXF format.

DwgCodePage

Changes the code page for text in drawings.

Accessing the Command

command bar: *dwgcodepage*
transparent: *'dwgcodepage*

: ***dwgcodepage***
Prompts you in the command bar:

This command sets DWGCODEPAGE to a new value. Characters will not be converted to unicode.

?=List/New code page <ANSI_1252> : ***(Enter a code page number, or type ?.)***

Code pages included with BricsCAD:

For Unix:
0: undefined
1: ascii $20-$7F,%%c,%%d,%%p Only
2: iso8859_1 Western Europe (Unix) $20-$FF,%%c,%%d,%%p
3: iso8859_2 Central Europe (Unix)
4: iso8859_3 Eastern Europe (Unix)
5: iso8859_4 Baltic (Unix)
6: iso8859_5 Cyrillic (Unix)
7: iso8859_6 Arabic (Unix)
8: iso8859_7 Greek (Unix)
9: iso8859_8 Hebrew (Unix)
10: iso8859_9 Turkish (Unix)

For DOS/Mac:
11: dos437 DOS USA
12: dos850 DOS Western Europe
13: dos852 DOS Eastern Europe
14: dos855 IBM Russian
15: dos857 IBM Turkish
16: dos860 DOS Portuguese
17: dos861 DOS Icelandic
18: dos863 DOS Canadian French
19: dos864 DOS Arabic
20: dos865 DOS Norwegian
21: dos869 DOS Greek
22: dos932 DOS Japanese
23: mac-roman Mac
24: big5 DOS Traditional Chinese
25: ksc5601 Korean Wansung
26: johab Korean Johab
27: dos866 DOS Russian
31: gb2312 DOS Simplified Chinese

For Windows:
28: ansi_1250 Win Eastern Europe
29: ansi_1251 Win Russian
30: ansi_1252 Win Western Europe (ANSI)
32: ansi_1253 Win Greek
33: ansi_1254 Win Turkish
34: ansi_1255 Win Hebrew
35: ansi_1256 Win Arabic
36: ansi_1257 Win Baltic
37: ansi_874 Win Thai
38: ansi_932 Win Japanese
39: ansi_936 Win Simplified Chinese GB
40: ansi_949 Win Korean Wansung
41: ansi_950 Win Tradissional Chinese big5
42: ansi_1361 Win Korean Johab
43: ansi_1200 Unicode (reserved)
--: ansi_1258 Win Vietnamese (reserved)

Command Options

Option	Description
? = List	Lists code pages available in the program.
New Code Page	Specifies the number of another code page.

DwgCompare

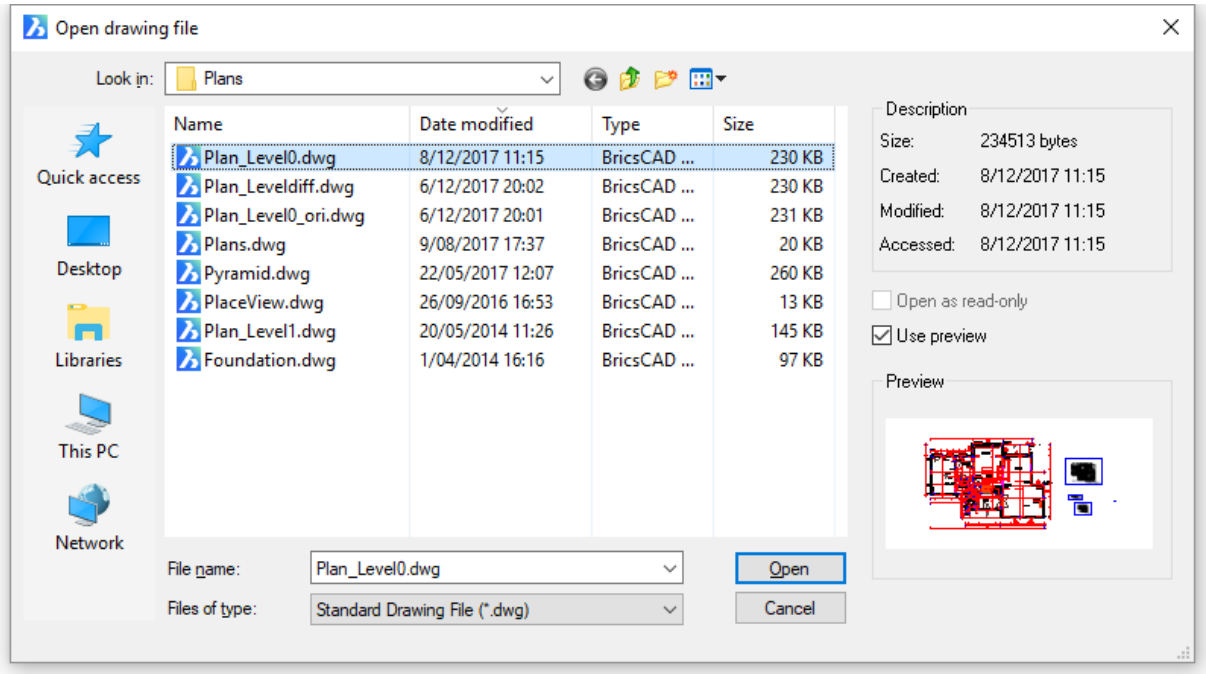
Compares a selected drawing with the current drawing during a comparison session.

Accessing the Command

command bar: *dwgcompare*
menu bar: *Tools | DWG Compare*
toolbar: *Compare |*

: ***dwgcompare***
Prompts you in the command bar:
CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]: *(Press Enter to open a File dialog box or choose the Limit option.)*

A File dialog box displays:



Select the drawing to compare the current drawing with, then click the Open button.
In the Structure panel, load the Default configuration file. See Load a configuration file.
Expand the Comparison node in the Structure panel:

NOTE You must execute the ENDCOMPARE command to exit the comparison session.

Command Options

Option	Description
Limit	Allows to set the value of the CMPDIFFLIMIT system variable, which sets the limit for comparison of entities. Values between 1 and 10000000 (ten million) are accepted. Prompts you: **New value for CMPDIFFLIMIT <10000000>** - type a value or press Enter to accept the current value.

Procedures

Setting the comparison system variables
Executing a comparison session
Analyzing the comparison in the Structure panel
Merging the compared drawings

Related Commands

3dCompare - executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel.
EndCompare - exits a drawing comparison setting, started by the DwgCompare command.
KeepMe - allows to merge the compared drawings visually.
StructurePanel - opens the Structure panel.

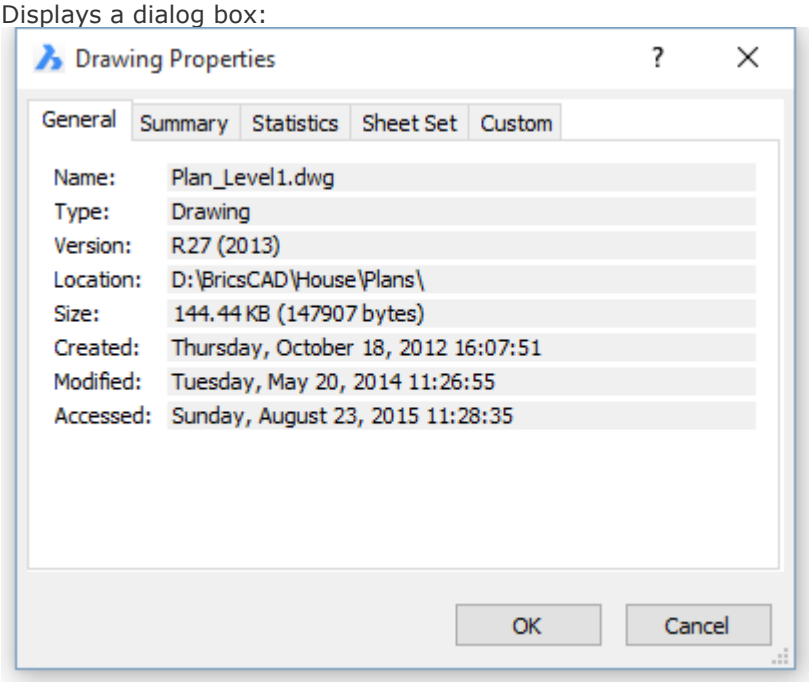
Dwgprops

Opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.

Accessing the Command

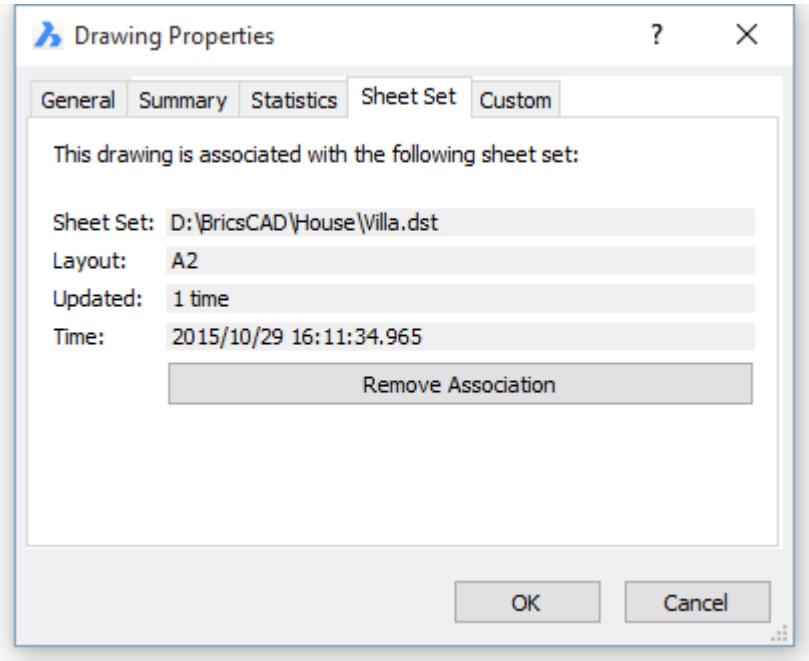
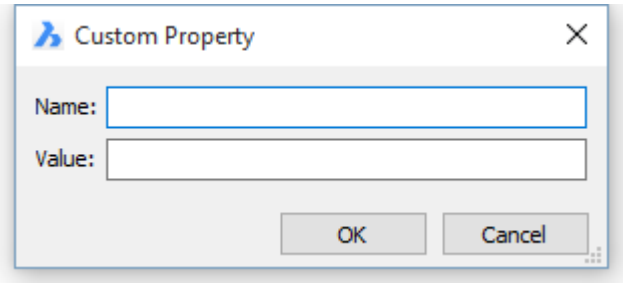
command bar: *dwgprops*
alias: none
menu bar: *File| Drawing Properties*

: ***dwgprops***
Displays a dialog box:



Command Options

Option	Description
General	Displays the *general properties* of the drawing.
Summary	Lets you define the *summary properties* of the drawing:
Statistics	Displays statistic information about the drawing.
SheetSet	Displays sheetset data if the drawing is associated with a sheetset. Click the *Remove Association* button to remove the association.

	
Custom	Displays the custom properties of the drawing. Click the *Add* button to add a new property.  Click the *Edit* button to edit the selected property. Click the *Delete* button to delete the selected property.

Procedures

To open the Drawing Properties dialog
Defining default properties
Adding custom properties

Related Commands

Field - inserts a field.
SecurityOptions - sets a password to protect a drawing
SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.
WhoHas - reports the ownership of a drawing file.

DxfIn

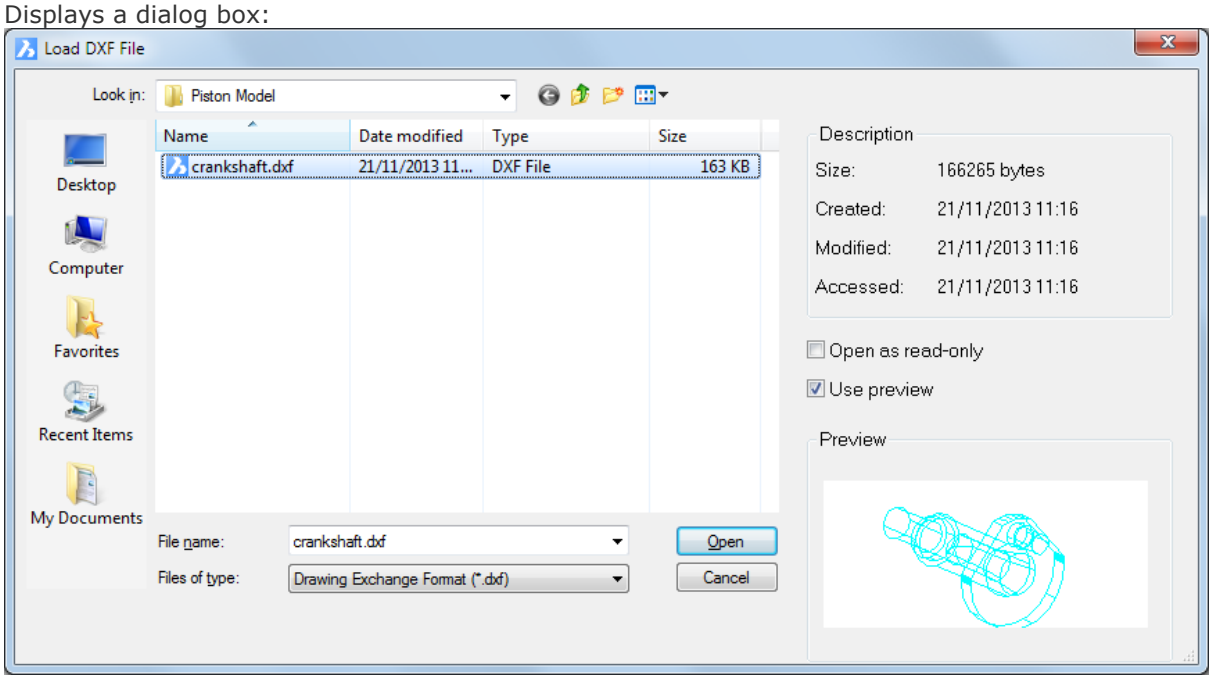
Imports DXF files (short for "drawing exchange format input").

Accessing the Command

command bar: *dxfin*

: ***dxfin***

Displays a dialog box:



*Choose a DXF file, and then click **Open**.*

Command Options

No options.

Related Commands

Export - exports drawings in a variety of formats.
DxfOut - exports drawings in DXF format.

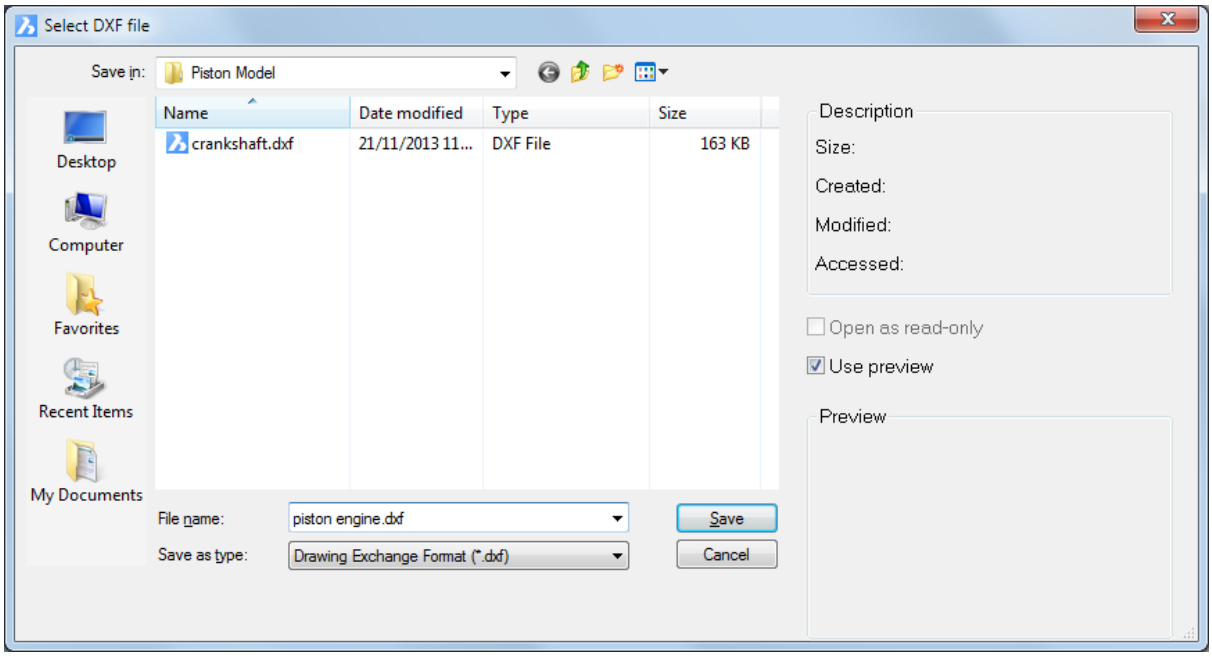
DxfOut

Exports drawings in DXF format (short for "drawing interchange format output").

Accessing the Command

command bar: *dxfout*
alias: *dx*

: ***dxfout***
Displays a dialog box:



Enter a file name, and then click Save.

Prompts you in the command bar:

Entities/Binary/Version/Enter decimal places of accuracy (0-16) <6>: ***(Enter an option.)***

Command Options

Option	Description
Entities	Exports only selected entities in the DXF file; prompts you: Select entities to export - choose the entities to export.
Binary	Exports the drawing in binary DXF format; otherwise, the DXF file is in ASCII (text) format.
Version	Specifies the DXF version number in which to save the drawing; prompts you: File version R2.5/R2.6/R9/R10/R11/R12/R13/R14/A2000/A2004/A2007 - enter a version number. Caution: exporting to an older version can cause some entities or properties to be lost.
Enter decimal places of accuracy	Specifies the accuracy of real numbers in the DXF file; enter a number between 0 (no decimal places) and 6.

	Some CNC (computerized numerical control) machines require 4 decimal places in DXF files.

Related Commands

Export - exports drawings in a variety of formats.
DwfOut - exports drawings in DWF format.

Dimensions

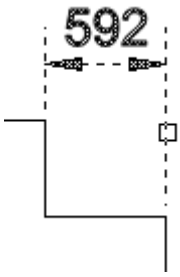
AiDimFlipArrow

Flips individual arrows of a dimension.

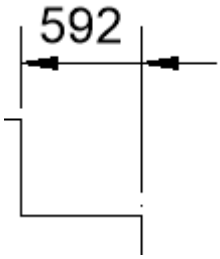
Accessing the Command

command bar: *aidimpfliparrow*
menu: *Dimension | Flip dimension arrowhead*
toolbar: *Dimensions |*

: *aidimpfliparrow*
Prompts you in the command bar:
Select entities: ***(Click the extension line of the dimension arrow you want to flip.)***



The arrow flips to the other side of the extension line.



Command Options

No options.

Related Commands

AiDimPrec - Changes the displayed precision of existing dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimAngular - places angular dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.

Dim

Places and edits dimensions at the 'Dimensioning command:' prompt.

Accessing the Command

command bar: *dim*
alias: *dimension*

: ***dim***
Prompts you in the command bar:
Dimensioning command
[HORizontal/VErtical/ALigned/ANgular/Leader/OBlique/ROtated/CEnter/Diameter/RAdius/Baseline/COntinue/ORdinate/Position/UPdate/STatus/OVerride/SEttings]: (Enter a dimension command.)

Dimensioning command: (Press Enter to exit the command.)

Command Options

Dimension Drawing Options	**Description**
Aligned	Places linear dimensions that are aligned to entities; see DimAligned.
Angular	Places angular dimensions that measure angles; see DimAngular.
Baseline	Places multiple linear and angular dimensions from the same base point; see DimBaseline.
Center	Places marks at the centers of circles and arcs; see DimCenter.
Continue	Continues linear and angular dimensions from the last endpoint; see DimContinue.
Diameter	Places diameter dimensions on circles and arcs; see DimDiameter.
Horizontal	Places horizontal linear dimensions; see DimLinear.
Leader	Places leaders; see DimLeader command.
Ordinate	Places x and y ordinate measurements from an origin point; see DimOrdiinate.
Radius	Places radial dimensions on circles and arcs; see DimRadius.
Rotated	Places linear dimensions at an angle; see DimLinear.
Vertical	Places vertical dimensions; see DimLinear.
Dimension Editing Commands	
Hometext	Resets dimension text to its original position after changes by TEdit; see DimEdit.
Newtext	Edits dimension text; see DimEdit.
Oblique	Changes the angle of extension lines; see DimEdit.
TEdit	Moves and reorients dimension text; see DimTEdit.
TRotate	Rotates dimension text; see DimTEdit.

Dimensioning Utility Commands	
Exit	Exits the Dim command; you can also press Esc or Enter.
Override	Overrides settings of dimension variables; see DimOverride.
Redraw	Redraws the current viewports; see Redraw.
Restore	Restores dimension variables to the settings originally specified by the dimension style; see -DimStyle.
Save	Saves the current settings of dimension variables as a new style; see -DimStyle.
Status	Reports the current values of dimension variables, including those overriding the current style; see -DimStyle.
Update	Updates the style of selected dimensions to that of the current dimension style; see -DimStyle.
Variables	Lists the values associated with the current dimension style; see -DimStyle.

Related Commands

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimArc - Places arc length dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

Dim1

Executes a single dimension command at the 'Dimensioning command:' prompt.

Accessing the Command

command bar: *dim1*

: ***dim1***
Prompts you in the command bar:

Dimensioning command
[HORizontal/VErtical/ALigned/ANgular/Leader/OBlique/ROtated/CEnter/Diameter/RAdius/Baseline/COntinue/ORdinate/Position/UPdate/STatus/OVerride/SEttings]: *(Enter a dimension command.)*

Command Options

Dimension Drawing Options	**Description**
Aligned	Places linear dimensions that are aligned to entities; see DimAligned.
Angular	Places angular dimensions that measure angles; see DimAngular.
Baseline	Places multiple linear and angular dimensions from the same base point; see DimBaseline.
Center	Places marks at the centers of circles and arcs; see DimCenter.
Continue	Continues linear and angular dimensions from the last endpoint; see DimContinue.
Diameter	Places diameter dimensions on circles and arcs; see DimDiameter.
Horizontal	Places horizontal linear dimensions; see DimLinear.
Leader	Places leaders; see DimLeader command.
Ordinate	Places x and y ordinate measurements from an origin point; see DimOrdiinate.
Radius	Places radial dimensions on circles and arcs; see DimRadius.
Rotated	Places linear dimensions at an angle; see DimLinear.
Vertical	Places vertical dimensions; see DimLinear.
Dimension Editing Commands	
Hometext	Resets dimension text to its original position after changes by TEdit; see DimEdit.
Newtext	Edits dimension text; see DimEdit.
Oblique	Changes the angle of extension lines; see DimEdit.
TEdit	Moves and reorients dimension text; see DimTEdit.
TRotate	Rotates dimension text; see DimTEdit.

Dimensioning Utility Commands	
Exit	Exits the Dim command; you can also press Esc or Enter.
Override	Overrides settings of dimension variables; see DimOverride.
Redraw	Redraws the current viewports; see Redraw.
Restore	Restores dimension variables to the settings originally specified by the dimension style; see -DimStyle.
Save	Saves the current settings of dimension variables as a new style; see -DimStyle.
Status	Reports the current values of dimension variables, including those overriding the current style; see -DimStyle.
Update	Updates the style of selected dimensions to that of the current dimension style; see -DimStyle.
Variables	Lists the values associated with the current dimension style; see -DimStyle.

Related Commands

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
DimArc - Places arc length dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimAligned

Draws dimensions parallel to (aligned with) selected entities; works with lines, polylines, arcs, and circles.

Accessing the Command

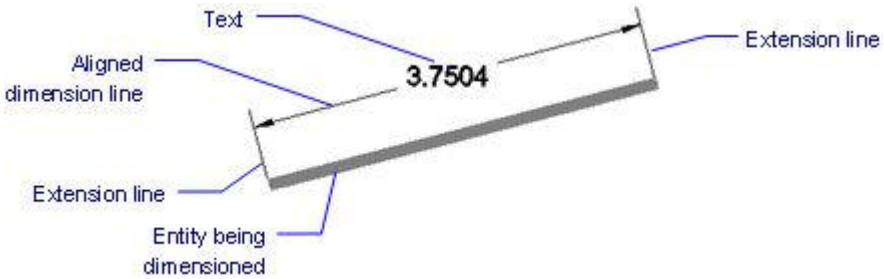
command bar: *dimaligned*
alias: *dal, dimali*
menu bar: *Dimension | Aligned*
toolbar: *Dimensions |*

: ***dimaligned***
Prompts you in the command bar:

Origin of first extension line <Select entity>: (Pick a point or press Enter to dimension an entity.)

Origin of second extension line: (Pick a point.)

Location of dimension line [Angle/Text]: (Pick a point, or choose an option.)



Command Options

Option	Description
Select entity	Dimensions the length of the selected entity. Prompts you: Select entity to dimension - select an entity. Lines, polyline segments, arcs, circles and edges of 3D solids in the XY-plane of the current UCS are accepted. Location of dimension line [Angle/Text] - pick a point or choose an option.
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines. Prompts you: Angle of dimension text - type a value.
Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD. Prompts you: Dimension text <xx.xx> - type a text or press Enter to accept the default text. Type <> in the dimension text to add the length of the dimension to the dimension text.

Procedures

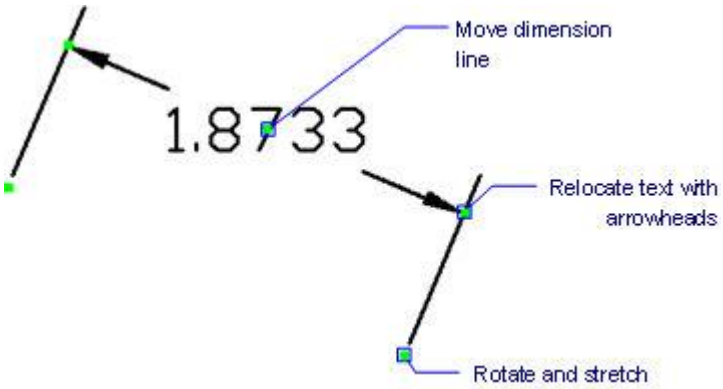
To create an aligned linear dimension

To create a horizontal or vertical linear dimension
To create a rotated linear dimension

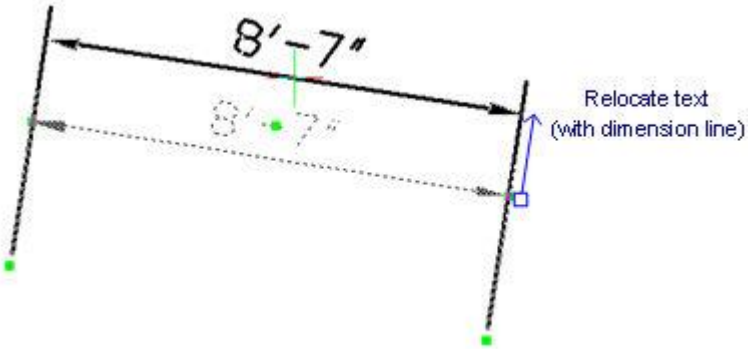
Grips Editing

Aligned dimensions can be edited directly through grips:

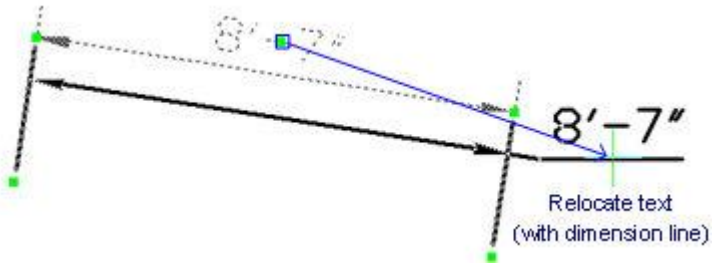
1. Select the aligned dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the dimension line to move the dimension line.
    - Drag the grip on the text to relocate it.
    - Drag either grip on extension lines to rotate and stretch the dimension.



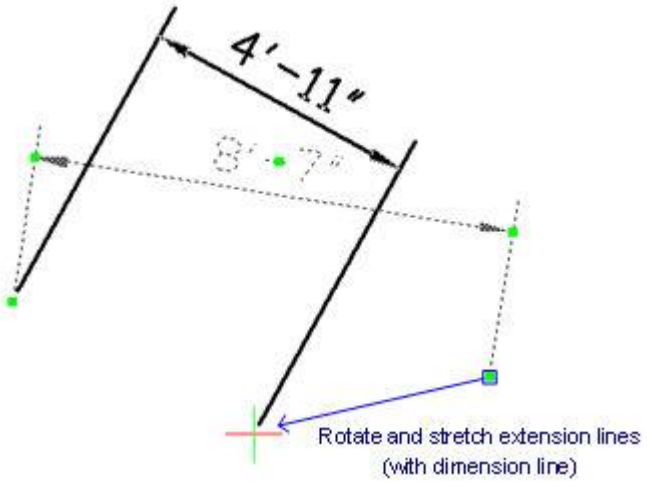
Move the dimension line by dragging either grip near the arrowhead:



Relocate the text and dimension line by dragging the grip centered on the dimension line (the leader is added automatically, if required):



Rotate and stretch the dimension by dragging either grip located at the base of the extension lines:



Related Commands

DimArc - Places arc length dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - changes the displayed precision of existing dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimAngular

Dimensions angles.

Accessing the Command

command bar: *dimangular*
alias: *dan, dimang*
menu bar: *Dimension | Angular*
toolbar: *Dimensions |*

: ***dimangular***

Prompts you in the command bar:

Select line, arc, or circle <Press ENTER to specify angle>: (Select a line, arc or circle; or press Enter to specify the angle vertex.)

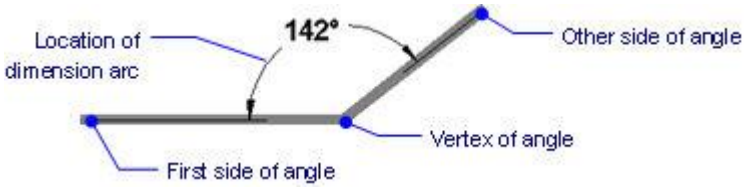
Vertex of angle: (Choose the vertex point.)

First side of angle: (Choose the starting point of the angle.)

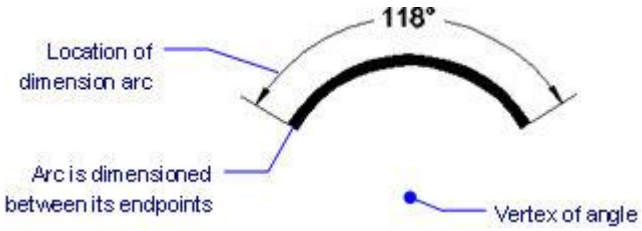
Other side of angle: (Choose the ending point of the angle.)

Angle/Text/<Location of dimension arc>: (Locate the dimension arc, or enter an option.)

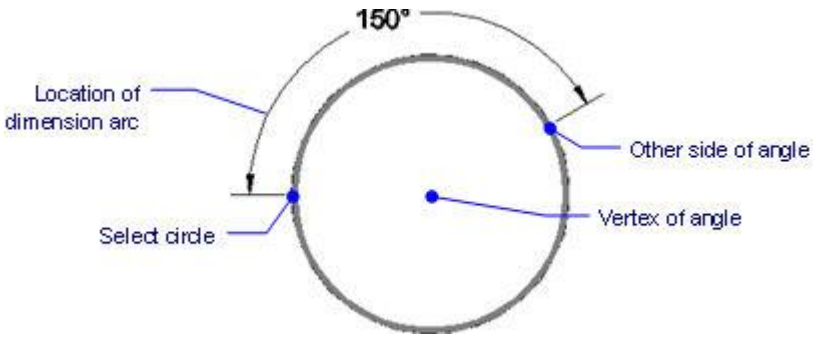
Dimension text = \A1;142%%d



Angular dimension between two lines
or a pair of polyline segments



Angular dimension of an arc



Angular dimension of two points on *a circle*

Command Options

Option	Description
Press ENTER to Specify Angle	Specifies the angular dimension by three points:
Vertex of Angle	First point is the vertex of the angle.
First Side of Angle	Second point is one leg of the angle.
Other Side of Angle	Third point is the second leg of the angle.
Location of Dimension Arc	Locates the position of the dimension arc.
Angle	Specifies the angle of the dimension text.
Text	Specifies the text, if different from the default text generated by BricsCAD.
Select Line, Arc, or Circle	Choose the entity to dimension -- line, polyline segment, arc, or circle: Line or polyline - you are prompted, "Other line for angular dimension." Circle - you are prompted, "Other side of angle." Arc - you are not prompted; BricsCAD goes immediately to 'Angle/Text/<Location of dimension arc>:' prompt.
Other Line for Angular Dimension	Choose a second line; the angular dimension is drawn between the two lines.
Other Side of Angle	Pick a second point on the circle; the angular dimension is drawn between the two points.

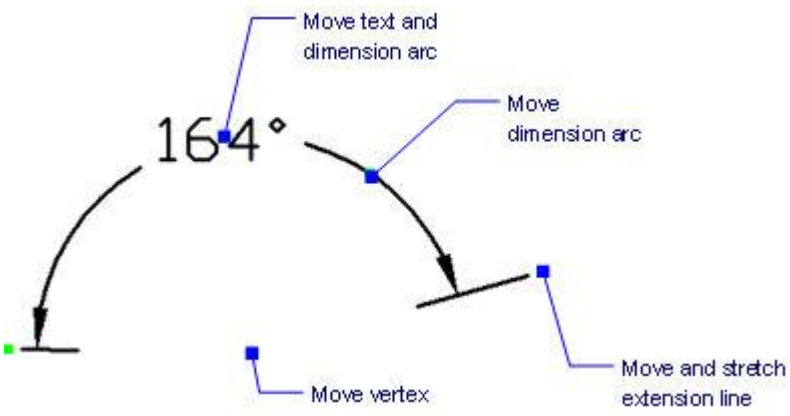
Procedures

To dimension an angle between two lines
To dimension an angle encompassed by an arc
To create an angular dimension defined by a vertex and two endpoints

Grips Editing

Angular dimensions can be edited directly through grips:

1. Select the angular dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag the grip on the text to relocate it.
    - Drag the grip on the dimension arc to move it.
    - Drag either grip on extension lines to move and stretch the dimension.
    - Drag the grip at the vertex to move it.



Related Commands

DimArc - Places arc length dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - changes the displayed precision of existing dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimArc

Places arc length dimensions.

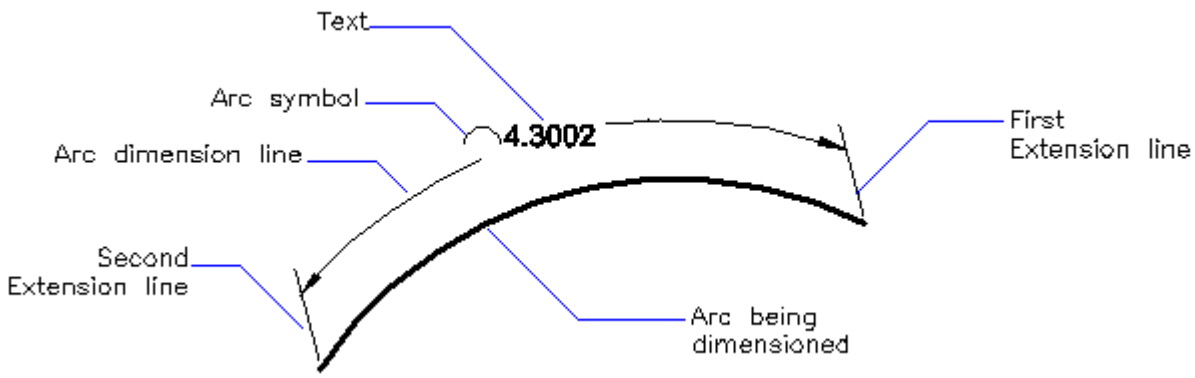
Accessing the Command

command bar: *dimarc*
alias: none
menu bar: *Dimension | Arc*
toolbar: *Dimensions |*

: ***dimarc***
Prompts you in the command bar:

Select arc or polyline arc segment: (click an arc or polyline arc segment)

Location of dimension arc [Angle/Text/Partial/Leader]: (Pick a point to locate the dimension arc away from the entity; or choose an option.)



Command Options

Option	Description
Angle	Specifies the angle of the text. Prompts you: Angle of dimension text - specify the rotation angle of the text.
Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD. Prompts you: Dimension text - specifies the replacement text.
Partial	Dimensions part of the selected arc or polyline arc segment. Prompts you: Specify first point for arc length dimension: specify the point on the arc or polyline segment to position the first extension line. Specify second point for arc length dimension: specify the point on the arc or polyline segment to position the second extension line.
Leader	Adds a leader. Prompts you: **Angle/Text/Partial/No leader/<Location of dimension arc>** - Pick a point to locate the dimension arc or choose an option.
No leader	Removes the leader. Prompts you: **Angle/Text/Partial/Leader/<Location of dimension arc>** - Pick a point to locate the dimension arc or choose an option.

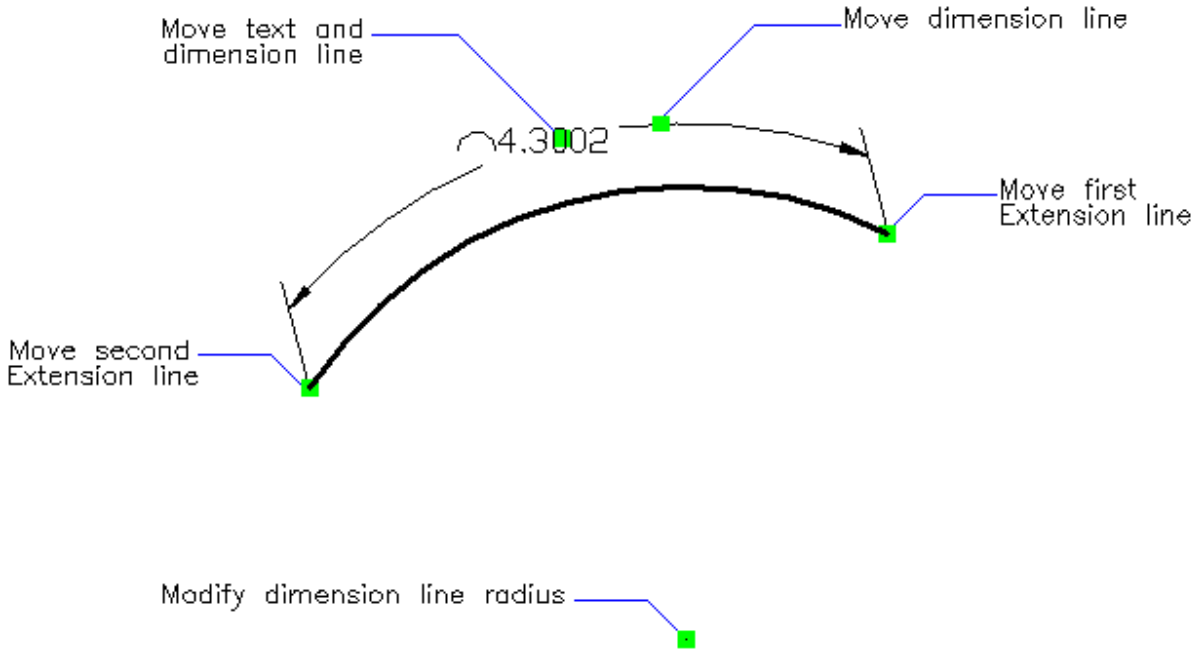
Procedures

To create an arc length dimension

Grips Editing

Arc length dimensions can be edited directly through grips:

1. Select the arc length dimension. Notice the grips.
2. Drag the grips to perform the following actions:



Related Commands

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.

DimOverride - overrides the values of the current dimension style.

DimBaseline

Places multiple linear and angular dimensions starting at the same base point; this command can only be used when at least one other dimension is already in the drawing.

Accessing the Command

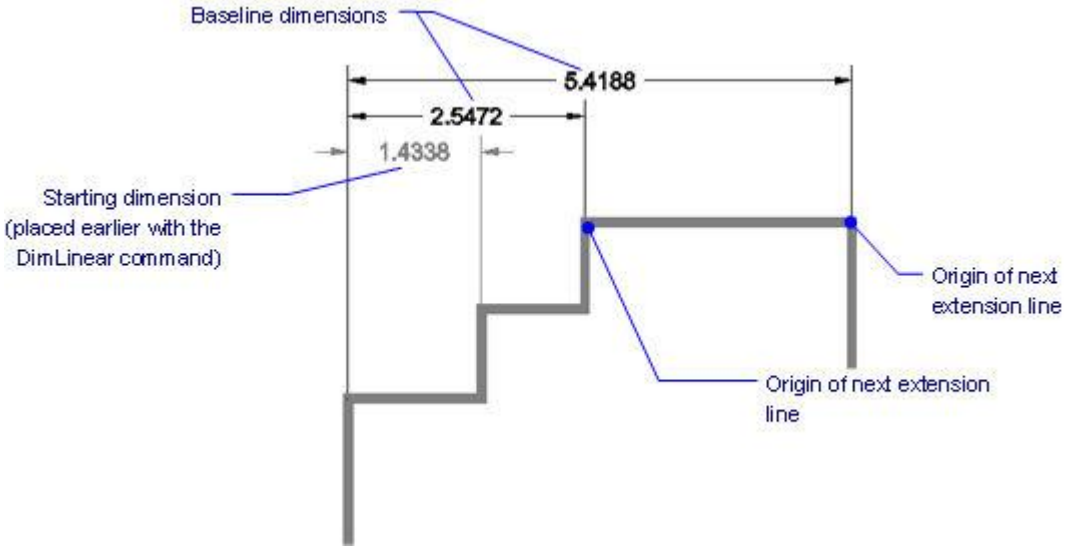
command bar: *dimbaseline*
aliases: *dba, dimbase*
menu bar: *Dimension | Baseline*
toolbar: *Dimensions |*

: ***dimbaseline***
Prompts you in the command bar:

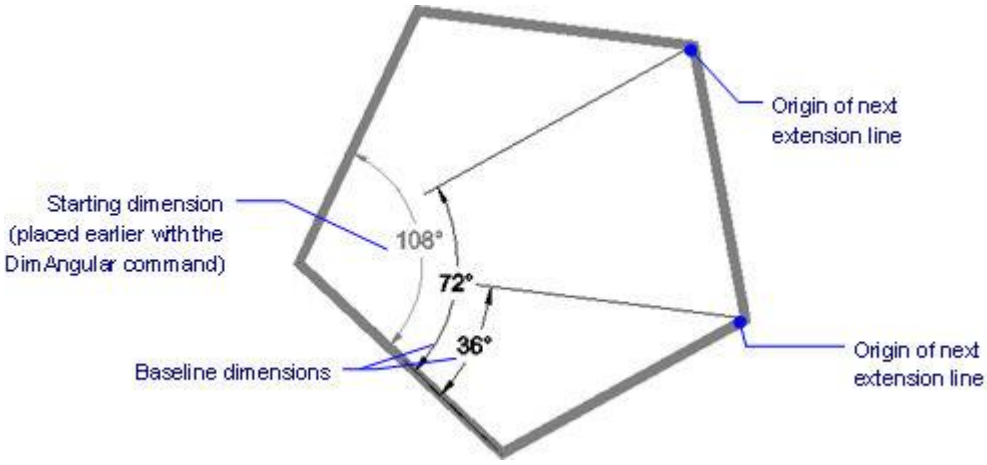
Baseline: ENTER to select starting dimension/<Origin of next extension line>: (Pick a point to locate the second extension line.)

Dimension text = \A1;2.5472

Baseline: ENTER to select starting dimension/<Origin of next extension line>: (Pick another point, or press Esc to exit the command.)



Baseline dimensions applied to a linear dimension



Baseline dimensions applies to an angular dimension

Command Options

Option	Description
ENTER to Select Starting Dimension	Chooses the linear or angular dimension from which to continue dimensioning.
Origin of Next Extension Line	Locates the endpoint of the second extension line; the first extension line is co-located with the first extension of the starting dimension. The command repeats this prompt until you press Esc.

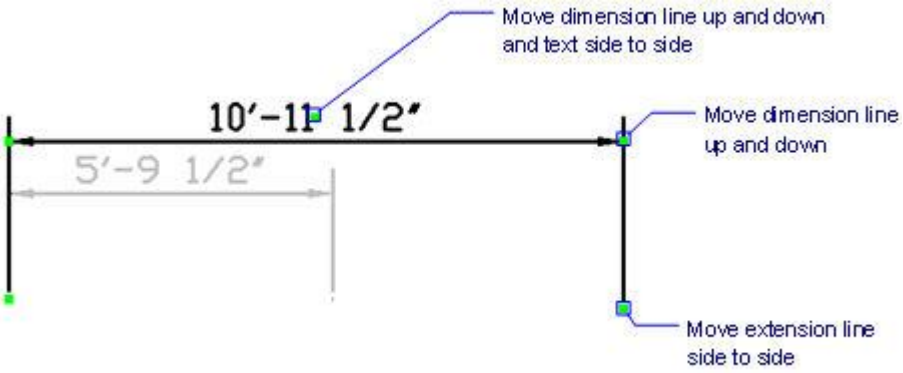
Procedures

To create baseline dimensions

Grips Editing

Baseline dimensions can be edited directly through grips:

1. Select the baseline dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the dimension line to move the dimension line; the extension lines change length.
    - Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
    - Drag either grip on the extension lines to move the extension line; the dimension line changes length.



Related Commands

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimBreak

Breaks dimension lines, extension lines and leaders at locations where they cross other entities.

Accessing the Command

command bar: *dimbreak*
menu bar: *Break Dimension*
toolbar: *Dimensions* |

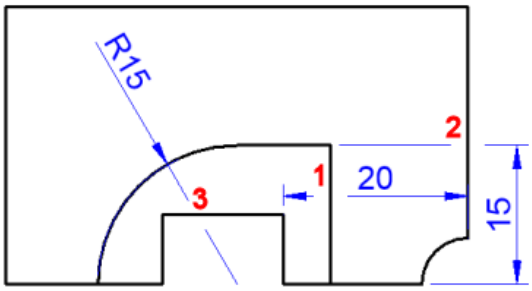
: ***dimbreak***
Prompts you in the command bar:

Select dimension to add/remove break or [Multiple]: (Select a dimension entity.)

Select object to break dimension or [Auto/Manual/Remove]: (Select an entity.)

Select object to break dimension: *(Select an entity or press Enter.)*

The dimension line and extension lines of the selected dimension are broken at the selected entity or entities. The size of the gap is defined by the Dim break size property of the current dimension style.



1. Dimension break on dimension line
2. Dimension break on extension line
3. Dimension break on leader

Command Options

Option	Description
Multiple	Allows to select multiple dimension entities. Prompts you: **Select dimensions to add/remove break:** Select a dimension entity. Entities in set: 1 **Select dimensions to add/remove break:** Select more dimension entities then press Enter to start selecting breaking entities.
Auto	The selected dimension(s) are broken at all intersections with other entities.
Manual	Allows to manually define the dimension break. You are prompted to specify two points.
Remove	Removes all dimension breaks from the selected dimension(s).

Related Commands

Break - removes a portion of an entity.

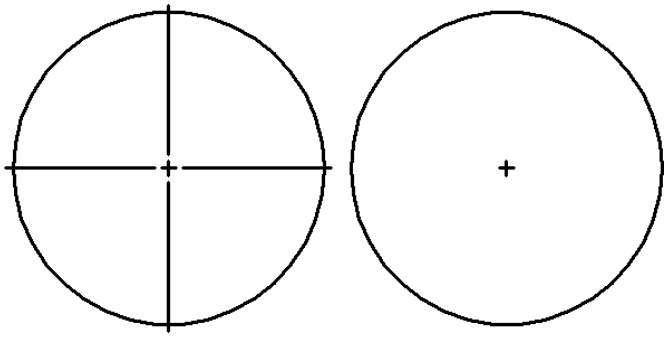
DimCenter

Places center marks at the center points of circles and arcs.

Accessing the Command

command bar: *DimCenter*
alias: *dce*
menu bar: *Dimension | Center Lines*
toolbar: *Dimensions | Circles |*

: ***dimcenter***
Prompts you in the command bar:
Select arc or circle to dimension: *(Choose an arc, polyline arc segemnt, or circle.)*



The DIMCEN system variable controls the length of center mark lines.
Left: DimCen = 0.09; right: DimCen = -0.09

Command Options

No Options.

Procedures

Creating center marks or center lines

Related Commands

CenterMark - creates a center mark entity associated with a selected circle or arc.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimContinue

Continues linear and angular dimensions from the endpoint of the previous dimension.

Accessing the Command

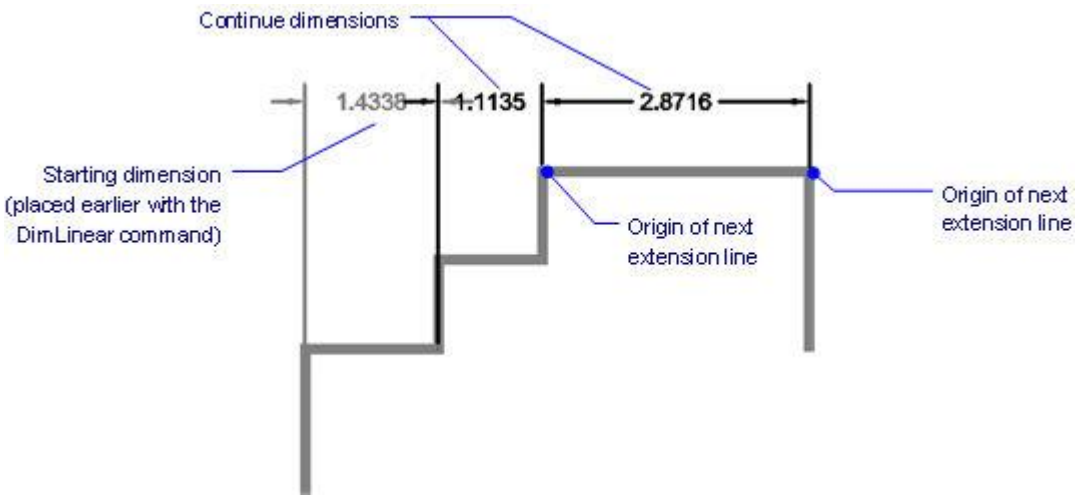
command bar: *dimcontinue*
aliases: *dco, dimcont*
menu bar: *Dimension | Continue*
toolbar: *Dimensions |*

: ***dimcontinue***
Prompts you in the command bar:

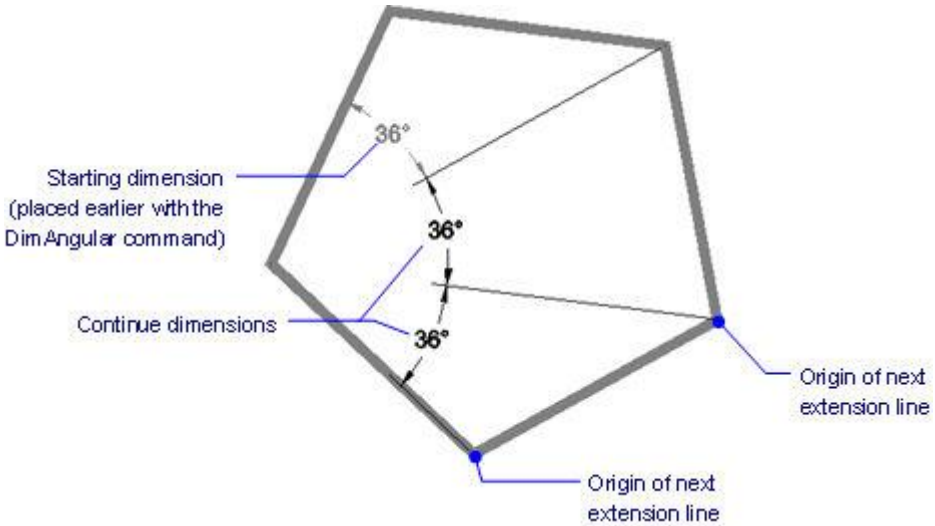
Continue: ENTER to select starting dimension/<Origin or next extension line>: (Choose a point for the endpoint of the second extension line.)

Dimension text = \A1;1.1135

Continue: ENTER to select starting dimension/<Origin or next extension line>: (Choose another point, or press Enter to exit command.)



Continue dimensions applied to a linear dimension



Continue dimensions applies to an angular dimension

Command Options

Option	Description
ENTER to Select Starting Dimension	Chooses the linear or angular dimension from which to continue dimensioning.
Origin of Next Extension Line	Locates the endpoint of the second extension line; the first extension line is co-located with the second extension of the previous dimension. The command repeats this prompt until you press Esc.

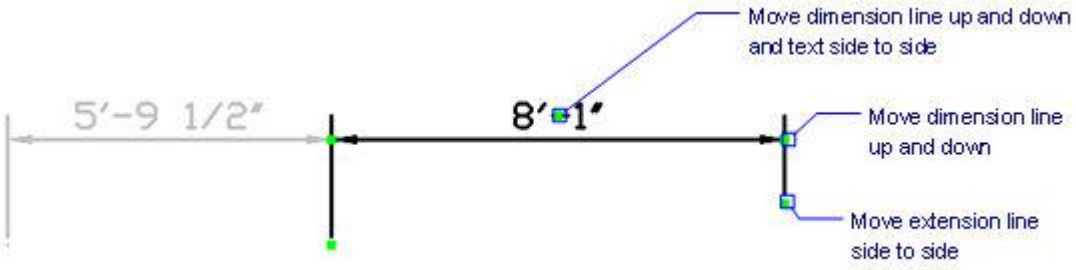
Procedures

To create continued dimensions

Grips Editing

Continued dimensions can be edited directly through grips:

1. Select the continued dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the dimension line to move the dimension line; the extension lines change length.
    - Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
    - Drag either grip on the extension lines to move the extension line; the dimension line changes length.



Related Commands

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimDiameter

Dimensions the diameter of circles and arcs, and places the center mark.

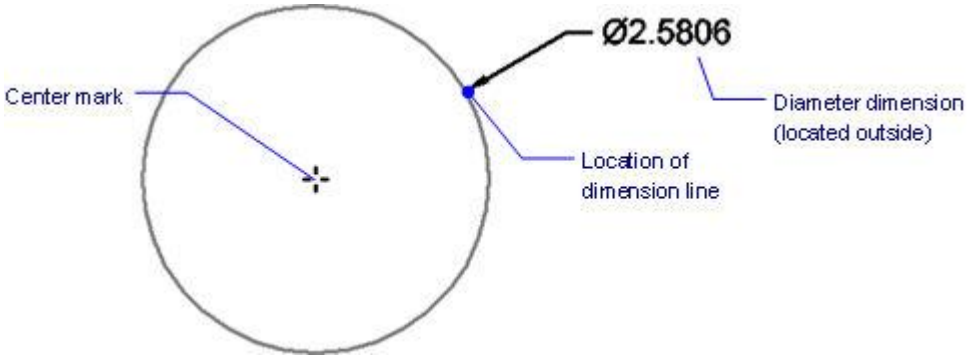
Accessing the Command

command bar: *dimdiameter*
aliases: *ddi, dimdia*
menu bar: *Dimension | Diameter*
toolbar: *Dimensions |* ∅

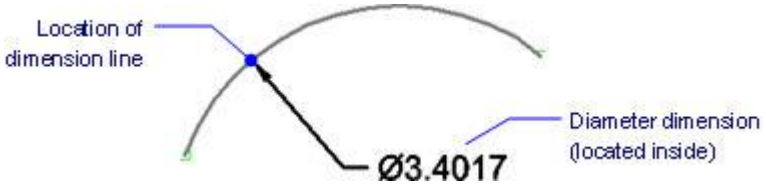
: ***dimdiameter***
Prompts you in the command bar:

Select arc or circle to dimension: (Choose an arc, polyline arc segment, or circle.)
Dimension text = xx.xx

Location of dimension line [Angle/Text]: (Position the dimension, or enter an option.)



A circle with diameter dimension placed outside



An arc with diameter dimension placed inside

Command Options

Option	Description
Angle	Changes the angle of the text.
Text	Changes the default text generated by BricsCAD.

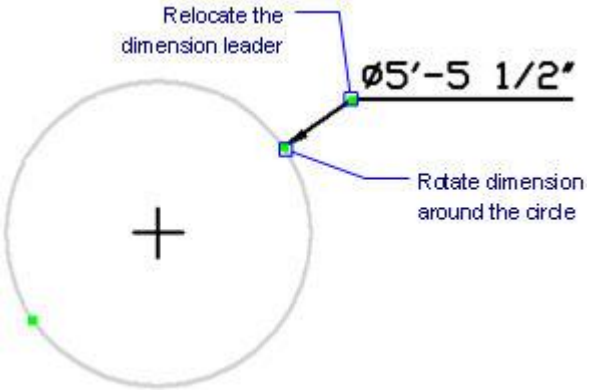
Procedures

To create a diametrical dimension
Creating center marks or center lines
To create a radial dimension

Grips Editing

Diameter dimensions can be edited directly through grips:

1. Select the diameter dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the circle to rotate the dimension around the circle.
    - Drag the grip on the leader to relocate it and the associated text.



Related Commands

DimArc - Places arc length dimensions.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - changes the displayed precision of existing dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimDisassociate

Removes the associativity from the selected dimension entities.

Accessing the Command

command bar: *dimdisassociate*
menu: *Dimension | Disassociate Dimensions*

: ***dimdisassociate***
Prompts you in the command bar:

Select dimensions to disassociate: (Select a dimension.)

Entities in set: 1

Select dimensions to disassociate: (Select more dimensions or right click to stop selecting.)

disassociated.

Command Options

No options.

Related Commands

DimReassociate - reassociates or associates dimensions to entities or points on entities.

DimEdit

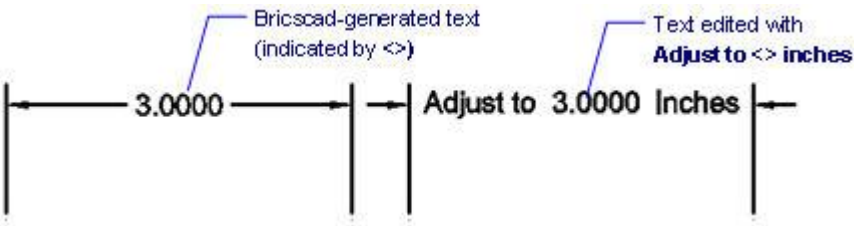
Changes the wording and angle of dimension text; changes the angle of extension lines.

Accessing the Command

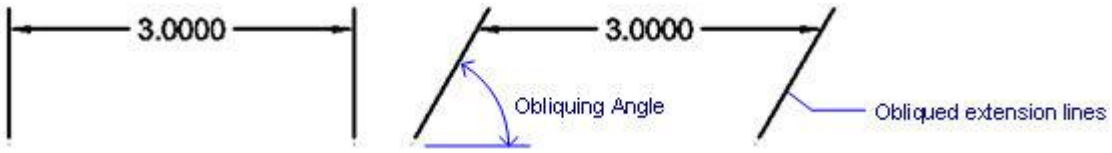
command bar: *dimedit*
aliases: *ded, dimed*
menu bar: *Dimension | Make Oblique*
toolbar: *Dimensions |*

: ***dimedit***
Prompts you in the command bar:

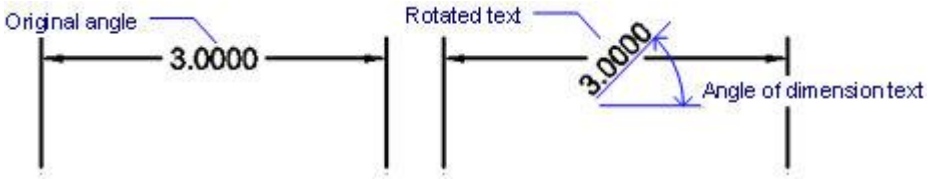
Edit dimension [Edit/Oblique/Rotate/REstore] <REstore text>: *(Enter an option, or press Enter to restore the dimension text to its default position.)*



Left: Original dimension text; right: Dimension text edited with the Edit Text option.



Left: Original extension lines; right: Extension lines set to an angle with the Oblique Lines option.



Left: Dimension text at original angle; right: Dimension text rotated with the Rotate Text option.

Command Options

Option	Description
Edit	Changes the text; prompts you: New dimension text <> - specify the new text; the <> symbol indicate the default text so that you can insert text in front or behind the default, such as "Adjust to <> inches." Selection dimensions to replace with new text - choose one or more dimensions; prompt repeats until you press Enter.
Oblique	Rotates (obliques or slants) the extension lines; useful for creating isometric dimensions. Prompts you:

	Select linear dimensions to make oblique - choose one or more linear dimensions; prompt repeats until you press Enter. Enter obliquing angle - specify the angle. The angle is measured counter clockwise from the positive x-axis.
Rotate	Rotates dimension text; prompts you: Angle of dimension text - specifies the angle. The angle is measured counter clockwise from the positive x-axis. Select entities to rotate text - choose one or more dimensions; prompt repeats until you press Enter.
REstore Text	Restores the text to its original position; this option does not restore edited text or obliqued extension lines. Prompts you: Select dimensions to restore text to default position - choose one or more dimensions; prompt repeats until you press Enter.

Procedures

Using grips to edit a linear dimension
Editing the dimension text
To rotate the dimension text
To reposition the dimension text
To restore the text position
To make the extension lines oblique

Related Commands

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - Changes the displayed precision of existing dimensions.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimLeader

Draws leaders.

Accessing the Command

command bar: *dimleader*
aliases: *lead, leader*
menu bar: *Dimension | Leader*
toolbar: *Dimensions |*

: ***dimleader***
Prompts you in the command bar:

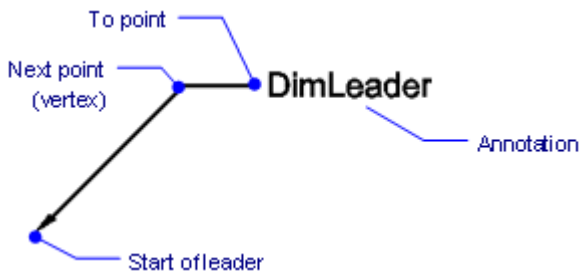
Start of leader: (Choose the starting point of the leader.)

Next point: (Choose the location of the first vertex.)

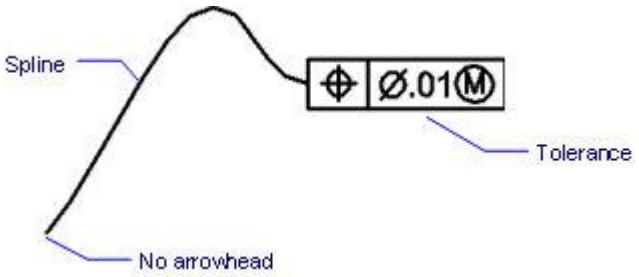
To point [Format/Undo/Annotation] <Annotation>: (Choose another vertex, enter an option, or press Enter to enter text.)

First line of annotation text <options>: (Enter text, or press Enter for annotation options.)

Next line of annotation text: (Enter additional text, or press Enter to exit the command.)



Leader drawn with arrowhead, straight line segments, and text annotation.



Leader drawn without arrowhead but with a spline segment, and tolerance annotation.

Command Options

Option	Description
Start of Leader	Locates the leader's arrowhead, and starts the leader line.
Next Point	Locates the first end point of the leader, or vertex.
To Point	Locates the next end point.
Format	Formats the leader line. Prompts you:

	[Arrow/None/SPline/STraight/Exit] <Exit> - choose an option or press Enter to exit the Format options. • Arrow: Draws the arrowhead. • None: Does not draw the arrowhead. • Spline: Draws the leader line as a spline. • Straight: Draws the leader line as straight segments. • Exit: Exits the Format options.
Undo	Undoes the last segment.
Annotation	Adds the leader's annotation. Prompts you: First line of annotation text <**options**> - Type the annotation text at the command prompt or press Enter to specify the annotation options. Next line of annotation text - Type the next line of the annotation text or press Enter to exit the command.

Options	Specifies the type of annotation.
	Prompts you:
	Dimension text options [Block/Copy/None/Tolerance/Mtext] <Mtext>: Select an option or press Enter to create an Mtext entity.
	• Block: Uses a block as the annotation; prompts you: ? to list blocks in drawing/~ to open the file dialog/<Block to insert>: type the name of a block in the drawing or type ? or ~
	  • ? : List the blocks in the current drawing.
	  • ~ : Displays a file dialog box, then choose a .dwg file to use as a block for the annotation.
	• Copy: Uses any other entity as the annotation; prompts you: Select an entity - Select an mtext, text, block reference or tolerance entity.
	• None: Leaves out the annotation.
	• Tolerance: Uses tolerance specification as the annotation; displays the Geometric Tolerance dialog box. See Tolerance command.
	• Mtext: Uses MText as the annotation; displays the Multiline Text dialog box. See MText command.

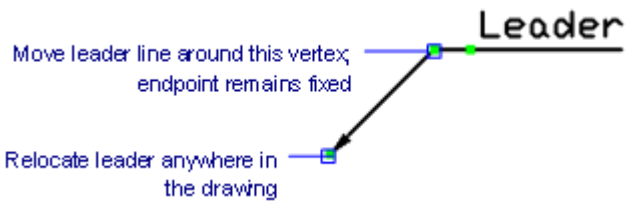
Procedures

To create a leader and an annotation
To create a leader only

Grips Editing

Leaders can be edited directly through grips:

1. Select the leader. Notice the grips.
2. Drag the grips to perform the following actions:
   - Drag a grip on a vertex of the leader line to stretch the line segments on either side of the vertex.
   - Drag the grip at the arrowhead to move the leader anywhere in the drawing. (The text is independent of the leader line.)



Related Commands

Leader - places leaders with text, blocks, entities, or tolerances.
QLeader - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.
MLeader - creates multileader entities using the current multileader style.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimLinear

Places linear dimensions horizontally, vertically, or rotated.

Accessing the Command

command bar: *dimlinear*
aliases: *dimlin, dli, dimhorizontal, dimrotated, dimvertical*
menu bar: *Dimension | Linear*
toolbar: *Dimensions |*

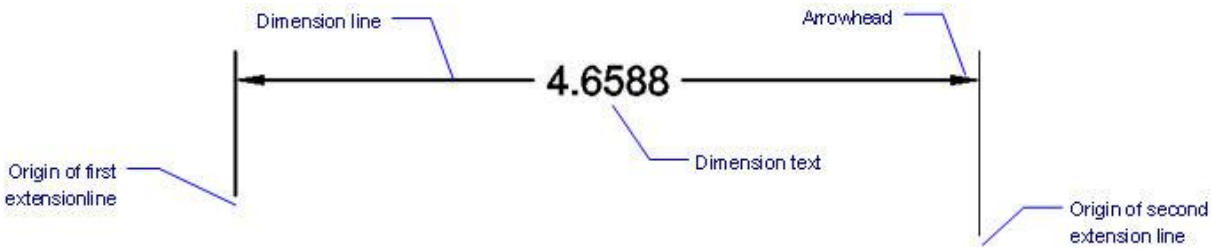
: ***dimlinear***

Prompts you in the command bar:

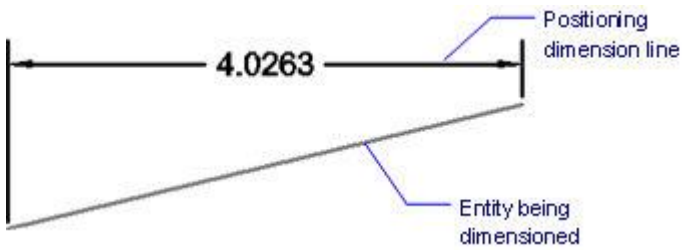
Origin of first extension line <Select entity>: (Pick a point; or press Enter to dimension an entity.)

Origin of second extension line: (Pick a point.)

[Angle/Text/Horizontal/Vertical/Rotated]: (Position the dimension line, or enter an option.)
Dimension text = xx.xx



Command Options

Option	Description
Select entity	Dimensions an entity. Prompts you: Select entity to dimension - Select a line, polyline segment, arc or circle to dimension; other entities cannot be dimensioned with this option. [**Angle**/**Text**/**Horizontal**/**Vertical**/**Rotated**] - Position the dimension line or choose an option. 
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines. Prompts you: Angle of dimension text - specify the rotation angle of the text.

Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD. Prompts you: Dimension text - specifies the replacement text.
Horizontal	Forces the dimension to be horizontal. prompts you: Location of dimension line [**Angle**/**Text**] - Position the dimension line or choose an option.
Vertical	Forces the dimension to be vertical. prompts you: Location of dimension line [**Angle**/**Text**] - Position the dimension line or choose an option.
Rotated	Rotates the dimension by a specified angle. Prompts you: Angle of dimension line <0.0> - specify the angle of the dimension line.

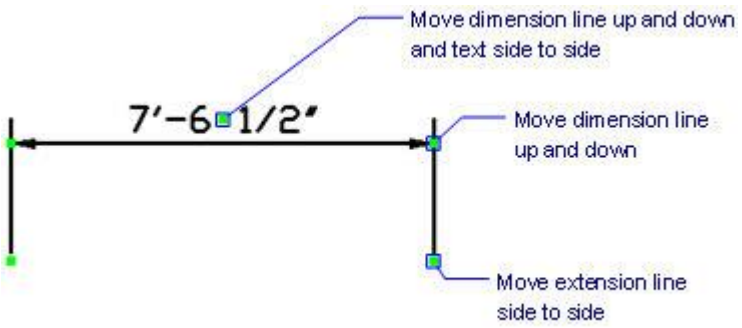
Procedures

To create a horizontal or vertical linear dimension
To create an aligned linear dimension
To create a rotated linear dimension

Grips Editing

Linear dimensions can be edited directly through grips:

1. Select the linear dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the dimension line to move the dimension line; the extension lines change length.
    - Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
    - Drag either grip on the extension lines to move the extension line; the dimension line changes length.



Related Commands

DimAligned - places aligned dimensions.
DimArc - Places arc length dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - changes the displayed precision of existing dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimOrdinate

Measures x and y ordinate distances from a common point at the current UCS origin.

Accessing the Command

command bar: *dimordinate*
aliases: *dor, dimord*
menu bar: *Dimension | Ordinate*
toolbar: *Dimensions |*

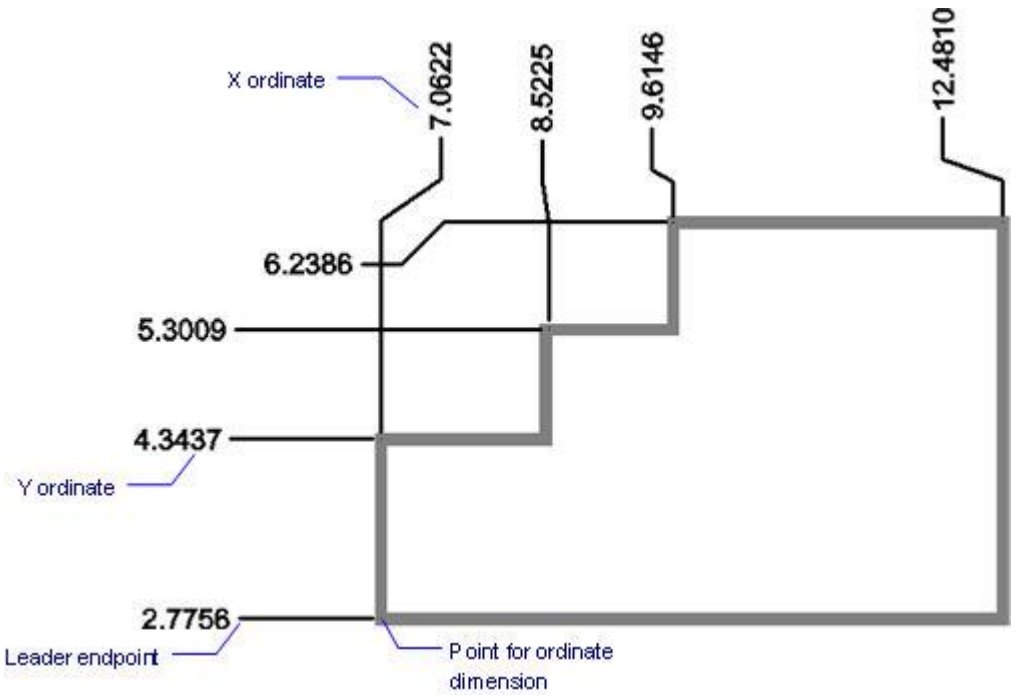
: ***dimordinate***
Prompts you in the command bar:

Select point for ordinate dimension: (Pick a point.)

Leader endpoint [Xdatum/Ydatum/Text/Angle]: (Pick an endpoint for the leader, or enter an option.)

Dimension text: = xx.xx

The distance is measured from the origin of the current UCS.



Command Options

Option	Description
X datum	Forces the ordinate dimension to measure the x distance; the distance is measured from the origin of the current UCS.
Y datum	Forces the ordinate dimension to measure the y distance; the distance is measured from the origin of the current UCS.
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines. Prompts you:

	Angle of dimension text - specify the rotation angle of the text.

Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD. Prompts you: Dimension text - specify the replacement text.

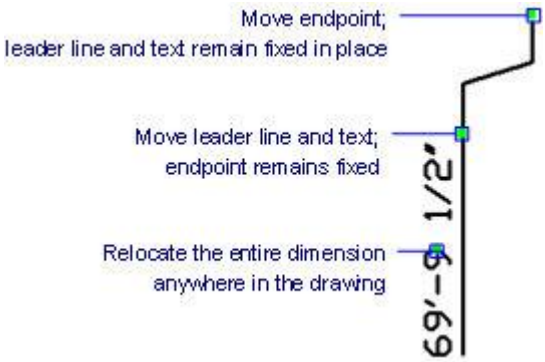
Procedures

To create an ordinate dimension

Grips Editing

Ordinate dimensions can be edited directly through grips:

1. Select an ordinate dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag the grip at the end point to relocate the endpoint; the leader and text remain fixed.
    - Drag the grip on the leader to relocate it and the associated text.
    - Drag the grip at the center of the text to relocate the entire dimension anywhere in the drawing.



Related Commands

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader - places leaders with text, blocks, entities, or tolerances.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

DimOverride

Overrides the values of the current dimension style.

Accessing the Command

command bar: *dimoverride*
aliases: *dor, dimover*

: ***dimoverride***
Prompts you in the command bar:

Select Dimension variable to override or [Clear]: (Enter the name of a dimension variable, or type C.)

DimVar is now <"xx.xx">. New value: (Enter new value.)

Dimension variable to override: (Enter the name of another dimension variable, or press Enter to continue.)

Select dimensions to use new value(s): (Choose one or more dimensions.)
Entities in set: 1

Select dimensions to use new value(s): (Choose more dimensions or press Enter to exit command.)

Command Options

Option	Description
Clear Overrides	Resets the values of dimension variables, clearing the overrides. rompts you: Select dimensions to clear overrides - choose the dimensions whose settings should be reset. Entities in set: 1 Select dimensions to clear overrides - choose more dimensions or press Enter to exit the command;

Related Commands

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.

DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimRadius

Dimensions the radii of arcs and circles.

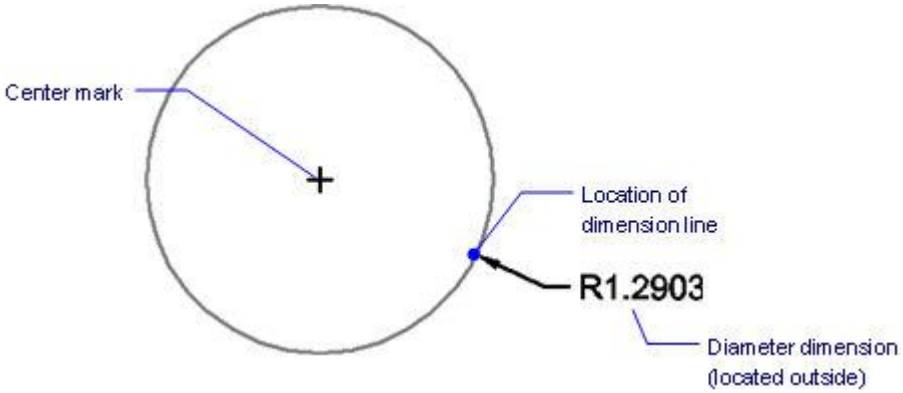
Accessing the Command

command bar: *dimradius*
alias: *dimrad, dra*
menu bar: *Dimension | Radius*
toolbar: *Dimensions |*

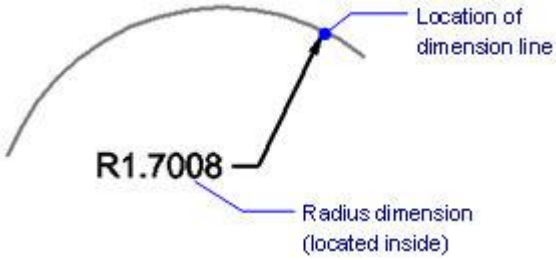
: ***dimradius***
Prompts you in the command bar:

Select arc or circle to dimension: **(Choose an arc, polyline arc, or circle.)**
Dimension text = xx.xx

Location of dimension line [Angle/Text]:: ***(Position the dimension, or enter an option.)***



A circle with radius dimension placed outside



An arc with radius dimension placed inside

Command Options

Option	Description
Angle	Specifies the angle of the text. Prompts you: Angle of dimension text - specify the rotation angle of the text.
Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD. Prompts you: Dimension text - specifies the replacement text.

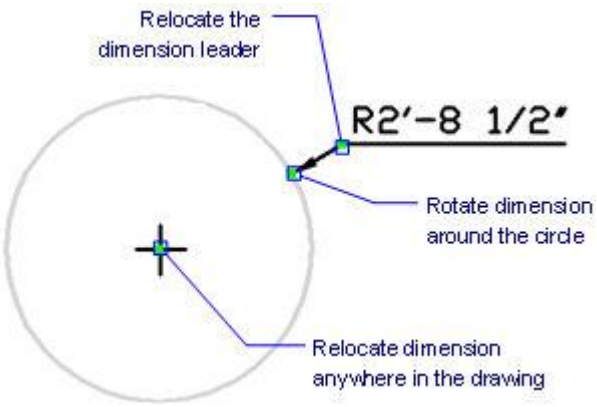
Procedures

To create a radial dimension
Creating center marks or center lines
To create a diametrical dimension

Grips Editing

Radial dimensions can be edited directly through grips:

1. Select the radius dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag the grip on the circle to rotate the leader and text around the circle.
    - Drag the grip on the leader to relocate it and the associated text.
    - Drag the grip at the center of the circle to relocate the dimension anywhere in the drawing.



Related Commands

DimArc - Places arc length dimensions.
DimDiameter - places diameter dimensions.
DimCenter - places center marks.
DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

AiDimFlipArrow - flips individual arrows of a dimension.
AiDimPrec - changes the displayed precision of existing dimensions.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.

DimOverride - overrides the values of the current dimension style.

DimReassociate

Reassociates or associates dimensions to entities or points on entities.

Accessing the Command

command bar: *dimreassociate*
menu: *Dimension | Reassociate Dimensions*

: ***dimreassociate***
Prompts you in the command bar:

Select entities or [Disassociated]: (Select one or more dimension entities or type D, then press Enter.)

Entities in set: 1

Select entities or [Disassociated]: (Select more dimension entities or type D, then press Enter or press Enter to stop selecting.)

Dimension entities in the selection set highlight one by one and you are prompted to specify association points or entities for the currently highlighted dimension entity.

A marker displays at the current position of the edited dimension point:

- Indicates a non-associated dimension point.
- Indicates an associated dimension point.

Depending on the dimension entity types in the selection set, the following prompts display:

Linear (rotated and aligned dimensions):

Specify first extension line origin or [Select entity] <next>: (Use an appropriate entity snap to specify the first extension line origin, type S then press Enter to select an entity or press Enter to skip to the next prompt.)

Specify second extension line origin <next>: (Use an appropriate entity snap to specify the second extension line origin or press Enter to skip to the next prompt.)

Diameter

Select arc, circle or polyline arc <next>: (Select an arc, circle or polyline arc segment or press Enter to skip to the next prompt.)

Radius

Select arc, circle or polyline arc <next>: (Select an arc, circle are polyline arc segment or press Enter to skip to the next prompt.)

Angular

2 Lines:

Select first line <next>: (Select a line or press Enter to skip to the next prompt.)

Select second line <next>: (Select a line or press Enter to skip to the next prompt.)

3 Points:

Specify angle vertex or [Select arc, circle or polyline arc] <next>: (Use an appropriate entity snap to specify the angle vertex or press Enter to skip to the next prompt.)

Specify first angle endpoint <next>: (Use an appropriate entity snap to specify the first angle endpoint or press Enter to skip to the next prompt.)

Specify second angle endpoint <next>: (Use an appropriate entity snap to specify the second angle endpoint or press Enter to skip to the next prompt.)

Arc or Circle:

Specify angle vertex or [Select arc, circle or polyline arc] <next>: (Type S, then press Enter or press Enter to skip to the next prompt.)

Select arc, circle or polyline arc <next>: (Select an arc, polyline arc segment or circle or press Enter to skip to the next prompt.)

Ordinate

Specify feature location <next>: (Use an appropriate entity snap to specify the new dimension point or press Enter to skip to the next prompt.)

Leader

Specify leader association point <next>: (Use an appropriate entity snap to specify the new leader association point or press Enter to skip to the next prompt.)

Command Options

Option	Description
Disassociated	Prompts to reassociate all disassociated dimension entities in the drawing. All disassociated dimensions highlight one by one.

Related Commands

DimDisassociate - removes the associativity from the selected dimension entities.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimAngular - places angular dimensions.
DimOrdinate - places x and y ordinate dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.

BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimRegen

Updates associative dimensions (short for "dimension regeneration").

Accessing the Command

command bar: *dimregen*

: ***dimregen***
Displays no prompt in the command bar.

Command Options

No options.

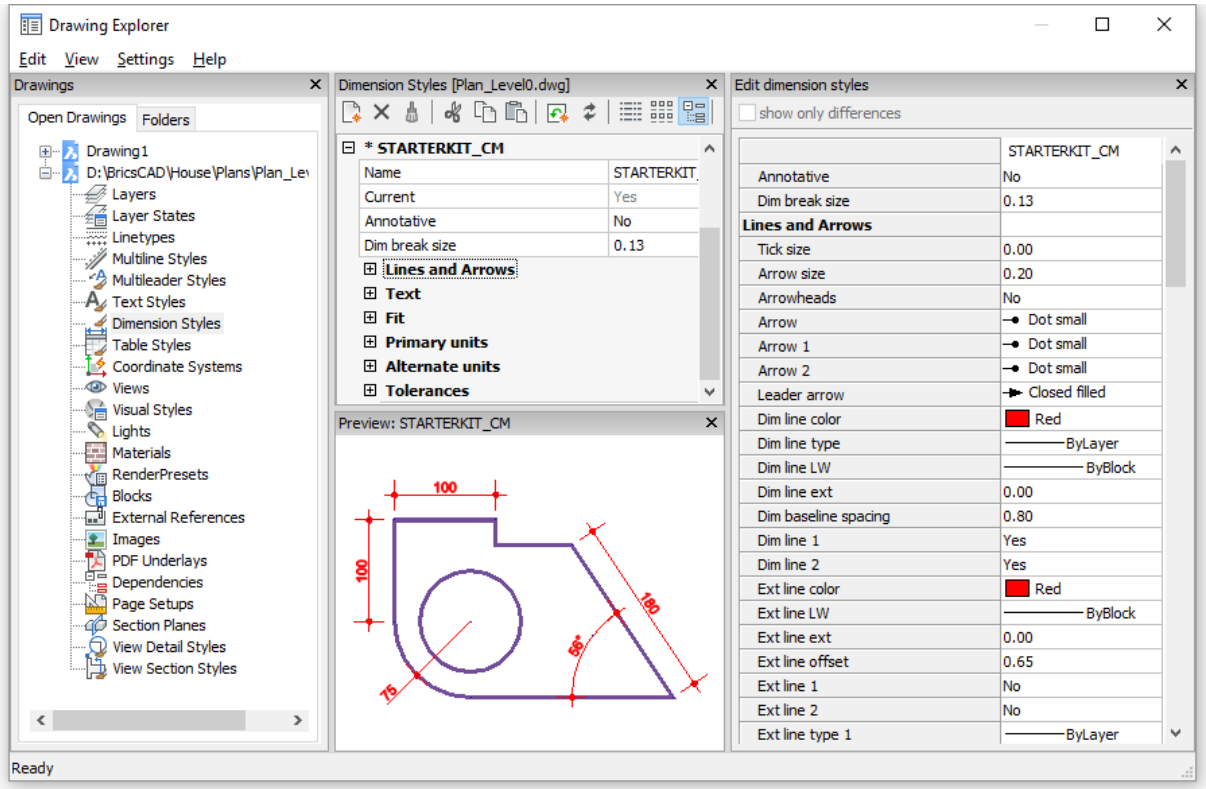
DimStyle

Creates and modifies dimension styles, through the Drawing Explorer.

Accessing the Command

command bar: *DimStyle*
aliases: *d, ddim, dimsty, ds, dst, expdimstyles, setdim*
menu bar: *Tools | Drawing Explorer | Dimension Styles...*
status bar: *Standard (or name of other current dimension style)*

: ***dimstyle***
Displays the Drawing Explorer at the Dimension Styles section:



Create or edit dimension styles, and then click X.

*Press **F1** to access help.*

<Overrides> **- allows dimension settings to be overriden, without creating a new style.**

*Standard **- lists the dimension settings for the dimension style named "Standard"; the * asterisk indicates that this is the current style.**

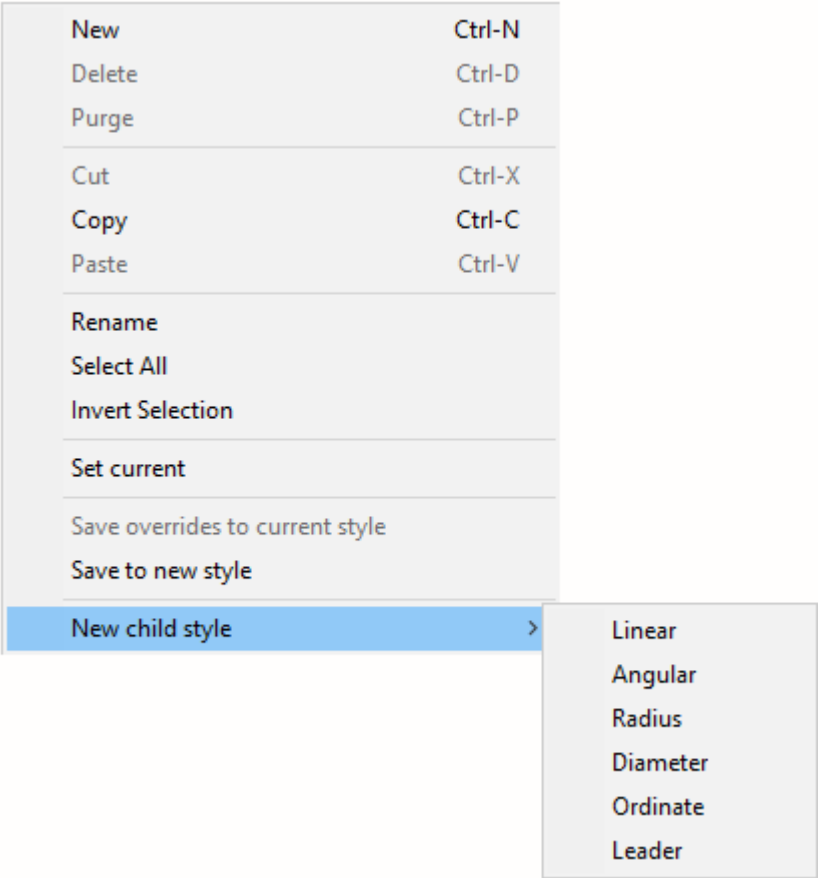
Command Options

Option	Description
Name	Names of the dimension style; all styles can be renamed and deleted, except for the Standard style.
Current	Indicates whether the style is current or not.
Annotative	Sets the annotative property of the dimension style.

Dim break size	Sets the value the gap created by the DimBreak command.
Show only differences	If multiple styles are selected, compares the selected styles and shows the differences only.

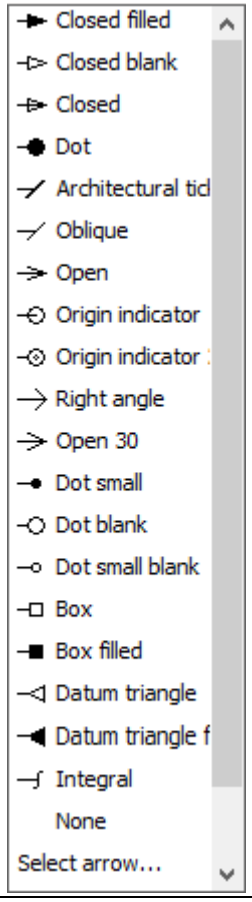
Dimension Child Styles

A dimension style can have up to 6 child styles: *Linear*, *Angular*, *Radius*, *Diameter*, *Ordinate* and *Leader*. If defined the child style will be used for the corresponding dimension types.
Right click the dimension style name, then choose New child style in the context menu and select a child style type.



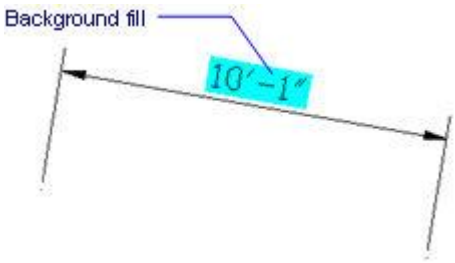
Dimension child styles share all settings with the parent style, except for the properties that are defined explicitly differently.

	Description
	Lines and Arrows |
	Tick size | 0 mm
	Arrow size | 180 mm
	Arrowheads | Yes
	Arrow | Closed filled
	Arrow 1 | Closed filled
	Arrow 2 | Closed filled
	Leader arrow | Closed filled
	Dim line color | ByBlock
	Dim line type | ByLayer
	Dim line LW | ByBlock
	Dim line ext | 100 mm
	Dim baseline spacing | 380 mm
	Dim line 1 | Yes
	Dim line 2 | Yes
	Ext line color | ByBlock
	Ext line LW | ByBlock
	Ext line ext | 180 mm
	Ext line offset | 180 mm
	Ext line 1 | Yes
	Ext line 2 | Yes
	Ext line type 1 | ByLayer
	Ext line type 2 | ByLayer
	Ext line fixed | No
	Ext line fixed length | 1 mm
	Center mark | Mark
	Center mark size | 90
	Arc length symbol | Preceding dimension tex
	Jog angle | 45
Arrow 1	Specifies the style of arrowhead for the starting end of the dimension line; choose from one of the following types:

	
Arrow 2	Specifies the style of arrowhead for the other end of the dimension line.
Arrow Size	Specifies the length of the arrowhead; the width is scaled proportionately.
Leader Arrow	Specifies the style of arrowhead for the starting end of the leader line.
Dim Line Color	Specifies the color of the dimension line; choose from: • Any Index Color. • Any True Color. • ByBlock. • ByLayer.
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.
Dim Line LW	Specifies the lineweight of the dimension line; choose from any lineweight supported by BricsCAD.
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; choose from: • Any Index Color. • Any True Color. • ByBlock. • ByLayer.
Ext Line type	Specifies the line type of both extension lines; choose from any line type loaded into the current drawing. To access additional linetypes, click Load, and then choose one from the Load Linetypes dialog box.
Ext Line LW	Specifies the lineweight of the extension line; choose from any lineweight supported by BricsCAD.
Ext Line Ext	Specifies the distance that the extension lines extend beyond the dimension line.
Ext Line Offset	Specifies the offset distance between the object and the start of the extension lines.
Ext Line 1	Toggles the display of the first extension line.
Ext Line 2	Toggles the display of the second extension line.
Ext line type 1	Specifies the line type of the first extension line; choose from any line type loaded into the current drawing. To access additional linetypes, click Load, and then choose one from the Load Linetypes dialog box.
Ext line type 2	Specifies the line type of the second extension line; choose from any line type loaded into the current drawing. To access additional linetypes, click Load, and then choose one from the Load Linetypes dialog box.
Ext line fixed length	Specifies the length of fixed-length extension lines.
Center Mark	Specifies the type of center mark: • Mark. • Line. • None.
Center Mark Size	Specifies the size of the center mark.
Arc length symbol	Specifies the location of the arc length symbol: • Preceding dimension text. • Above dimension text. • Not displayed.
Jog angle	Specifies the angle of the jog; default is 45 degrees. Enter another angle.

	Description
Text	
Text style	Standard
Text color	ByBlock
Text fill	No fill
Text fill color	ByBlock
Text height	180 mm
Draw frame around text	No
Text position vertical	Above
Text position horizontal	Centered
Text offset	90
Text vertical offset	180
Text inside align	No
Text outside align	Yes
Text view direction	Left to right

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; choose from: • Any Index Color. • Any True Color. • ByBlock. • ByLayer.
Text Fill	Determines whether the dimension text has a rectangular background filled with color.  Choose from: • No fill - does not apply a background fill. • Background - uses the background color of the drawing, usually white or black. • Color - uses the color specified by the Text Fill Color option.
Text Fill Color	Specifies the background fill color when Text Fill is set to Color. Choose a color from the drop list, or click Select Color to access the Select Color dialog box.
Text Height	Specifies the height of the text.
Draw Frame Around Text	Toggles a rectangle around the text.
Text Pos Vert	Justifies the text vertically relative to the dimension line: • Centered - centers the text on the dimension line.

	• Above - places the text above the dimension line. • Outside - places text outside of the extension lines. • JIS - generally places text above the dimension line, according to the Japanese Industry Standard.
Text Pos Hor	Justifies the text horizontally relative to the extension lines: • Centered - centers the text between the extension lines. • First Extension Line - places the text near the first extension line. • Second Extension Line - places the text near the second extension line. • Over First Extension - places the text over the first extension line. • **Over Second Extension -** places the text over the second extension line.
Text Offset	Specifies the size of gap between the dimension line and the text.
Text Inside Align	Justifies text when located between extension lines: • Aligned with Dimension Line - aligns text with dimension line. • Horizontal - forces text to be always horizontal.
Text Outside Align	Justifies text when located outside of the extension lines: • Aligned with Dimension Line - align text with dimension line. • Horizontal - forces text to be always horizontal.

	Description
Fit	
Arrow and text fit	Best fit
Text inside	No
Dim line inside	No
Text movement	Keep dim line with text
Dim scale overall	0.1
Place text manually	No
Dim line forced	No
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 BricsCAD to figure out where to place text and arrows, depending on the space between extension lines.
Text Inside	Toggles whether the text is forced between extension lines: • On - forces text between extension lines. • Off - draws text outside, if insufficient space between extension lines.
Dim Line Inside	Toggles whether the dimension line is forced between extension lines: • On - forces dimension line always between extension lines.

	• Off - draws dimension line outside, if insufficient space between extension lines.
Text Movement	Specifies what happens when text is moved away from its default location: • Keep dim line with text - moves dimension line with the text, stretching the extension lines. • Move text, add leader - draws a leader between the text and the dimension line. • Move text, no leader - does not draw a leader.
Dim Scale Overall	Specifies the overall scale factor for dimensions; this affects the size of arrows and text only. This setting is not editable for annotative dimension styles. It is recommended to set Dim Scale Overall = 1 for annotative dimension styles.
Place Text Manually	Toggles whether the user must always specify the text location when creating dimensions.
Dim Line Forced	Forces the dimension line to be always drawn; forces leaders to be drawn with the DimDiameter and DimRadius commands.

	Description
	Primary units |
	Dim units | Decimal
	Dim precision | 0
	Fractional type | Horizontal
	Decimal separator | .
	Dim round | 0
	Dim prefix |
	Dim suffix |
	Dim sub-units suffix |
	Dim scale linear | 1
	Dim sub-units scale | 100
	Suppress leading zeros | No
	Suppress trailing zeros | No
	Suppress zero feet | Yes
	Suppress zero inches | Yes
	Dim angle units | Decimal degrees
	Dim angle precision | 0.0000
	Suppress angle leading zeros | No
	Suppress angle trailing zeros | No
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 - stack stacks diagonally, with a diagonal separator line. • None - does not stack fractions, placed horizontally with a slash separator.
Decimal Separator	Specifies the character used to indicate the decimal point; can be any character. North American countries use the period; European countries use the comma.
Dim Round	Specifies the rounding of decimal numbers; range is none to 8 decimal places.
Dim Prefix	Specifies prefix text that appears in front of the dimension text, if any.
Dim Suffix	Specifies suffix text that appears after the dimension text, if any.
Dim sub-units suffix*	Specifies suffix text in case the sub unit is used. For example, enter 'cm' to display .96m to display as 96cm.
Dim Scale Linear	Specifies the multiplier for dimension values, such as 25.4 for changing inches to millimeters.
Dim sub-units scale*	Sets the scale factor for the sub-units. E.g. enter 10 if the drawing unit is cm, and the sub-unit suffix is mm.
Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point; for example, 0.23 is displayed as: • On - .23 • Off - 0.23
Suppress Trailing Zeros	Toggles the display of zeros in front of the decimal point; for example, 1.2300 is displayed as: • On - 1.23 • Off - 1.2300
Suppress Zero Feet	Toggles the display of zero feet; for example, 0'-3" is displayed as: • On - 3" • Off - 0'-3"
Suppress Zero Inches	Toggles the display of zero inches; for example, 1'-0" is displayed as: • On - 1' • Off - 1'-0"
Dim Angle Units	Specifies the format of units in angular dimensions: • Decimal Degrees - 360 degrees in a circle; for example, 123.45 degrees • Deg/Min/Sec - degrees, minutes, seconds; for example, 123d 12' 45.67". • Grads - 400 grads in a circle; for example, 230g. • Rads - 2pi radians in a circle; for example, 1.5r.
Dim Angle Precision	Specifies the number of decimal places; ranges from 0 to 8.
Suppress Angle Leading Zeros	Toggles the display of zero degrees; for example, 0.1234 degrees is displayed as: • On - .1234 • Off - 0.1234

Suppress Angle Trailing Zeros	Toggles the display of zeros after degrees; for example, 0.1200 degrees is displayed as: • On - 0.12 • Off - 0.1200

* Suppress leading zeros must be set to *Yes* to display dimension distances less than one unit in sub units

	Description
Alternate units	
Alt enabled	No
Alt units	Architectural Stacked
Alt precision	0'-0 1/4"
Alt scale factor	0.03937
Alt sub-units scale	100
Alt round	0
Alt prefix	
Alt suffix	
Alt sub-units suffix	
Alt suppress leading zeros	No
Alt suppress trailing zeros	No
Alt suppress zero feet	Yes
Alt suppress zero inches	Yes

Alternate Enabled	Toggles the display of alternative units: • On - second, alternative units displayed to the right of primary units. • Off - only primary units are displayed.
Alt Units	Specifies the display units for dimensions: • Scientific - scientific notation, such as 1.2345E+01. • Decimal - metric, such as 1.2345. • Engineering - feet and decimal inches, such as 1'-2.3456". • Architectural - feet and fractional inches, such as 1'-2 1/16". • Fractional - fractional inches only; no feet, such as 14 1/16". • Windows Desktop - uses the units set by Windows.
Alt Precision	Specifies the precision of units, either decimal places or fractional accuracy.
Alt Scale Factor	Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).
Alt Round	Specifies the rounding of alternate numbers; range is none to 8 decimal places.
Alt Prefix	Specifies prefix text that appears in front of the alternate text, if any.
Alt Suffix	Specifies suffix text that appears in after the alternate text, if any.
Alt Leading Zeros	Toggles the display of zeros in front of the decimal point; for example, 0.23 is displayed as: • On - .23 • Off - 0.23
Alt Trailing Zeros	Toggles the display of zeros in front of the decimal point; for example, 1.2300 is displayed as: • On - 1.23 • Off - 1.2300
Alt Zero Feet	Toggles the display of zero feet; for example, 0'-3" is displayed as: • On - 3" • Off - 0'-3"

Alt Zero Inches	Toggles the display of zero inches; for example, 1'-0" is displayed as: • On - 1' • Off - 1'-0"

	Description
Tolerances	
Tolerance display	No
Limits display	No
Tolerance precision	0.0000
Tolerance limit lower	0
Tolerance limit upper	0
Tolerance text height	1 mm
Tolerance position vertical	Middle
Tolerance suppress leading zeros	No
Tolerance suppress trailing zeros	No
Tolerance suppress zero feet	Yes
Tolerance suppress zero inches	Yes
Alt tolerance precision	0'-0 1/4"
Alt tolerance suppress leading zeros	No
Alt tolerance suppress trailing zeros	0
Alt tolerance suppress zero feet	Yes
Alt tolerance suppress zero inches	Yes

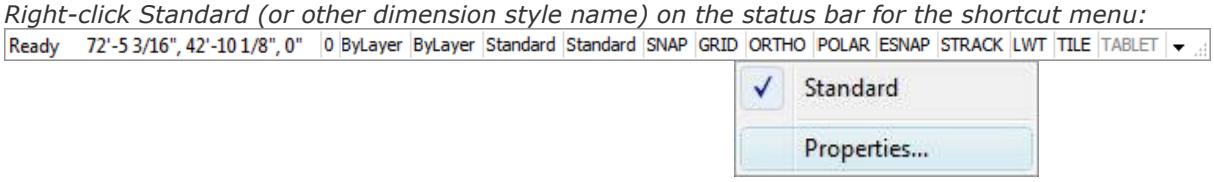
Tolerance Display	Toggles the display of tolerance text.
Tolerance Precision	Specifies the display precision of tolerance text; range is 0 to 8 decimal places or 1/1 to 1/256 inch.
Tolerance Limit Lower	Specifies the value of the upper tolerance.
Tolerance Limit Upper	Specifies the value of the lower tolerance.
Tolerance Text Height	Specifies the height of tolerance text.
Tolerance Pos Vert	Locates the tolerance text relative to the dimension text: • Bottom - aligns tolerance text with the bottom of the dimension text. • Middle - aligns tolerance text with the middle of the dimension text. • Top - aligns tolerance text with the top of the dimension text.
Tolerance Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point.
Tolerance Suppress Trailing Zeros	Toggles the display of zeros behind the decimal point.
Tolerance Suppress Zero Feet	Toggles the display of zero feet.
Tolerance Suppress Zero Inches	Toggles the display of zero inches.
Alt Tolerance Precision	Specifies the number of decimal places for tolerances in alternate units.
Alt Tolerance Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point.

Alt Tolerance Suppress Trailing Zeros	Toggles the display of zeros behind the decimal point.

Alt Tolerance Suppress Zero Feet	Toggles the display of zero feet.
Alt Tolerance Suppress Zero Inches	Toggles the display of zero inches.

Status Bar Menu

Right-click Standard (or other dimension style name) on the status bar for the shortcut menu:



Shortcut Menu	Description
Standard	Chooses the default dimension style. When the drawing contains additional dimension styles, their names are listed here.
Properties	Opens the Dimension Styles section of the Drawing Explorer.

Procedures

Exploring dimension styles
Comparing dimension styles
To list the current status of the dimension variables
To create a new dimension style
To edit a dimension style
To define dimension style overrides
To rename a dimension style
To set a dimension style current
Using the status bar to set a dimension style current
To copy a dimension style between drawings
To move a dimension style between drawings
To delete a dimension style

Related Commands

AiDimStyle - creates Dimension Styles based on the properties of selected Dimension, Tolerance or Leader entities.
-Dimstyle - creates and modifies dimension styles at the command line.
Explorer - opens the Drawing Explorer dialog box.

-DimStyle

Creates and modifies dimensions styles at the command line.

Accessing the Command

command bar: *-dimstyle*
alias: *-dst*
status bar: *Standard*

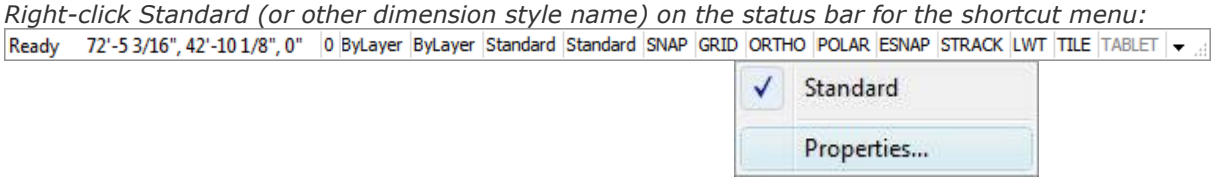
: ***-dimstyle***
Prompts you in the command bar:
Current dimension style: Standard Annotative: No
Dimension styles or [?/ANnotative/Apply/Save/STatus/Variables/Restore] <Restore>: *(Enter an option.)*

Command Options

Option	Description
?	Lists the names of all dimension styles loaded into the current drawing, other than Standard. Prompts you: Dimension style(s) to list <*> - enter a name or press Enter for all.
Annotative	Sets the annotative property of the dimension style.
Apply	Changes or restores the style of existing dimensions by applying the current style. Prompts you: Select dimensions to apply the current style - choose one or more dimensions.
Save	Saves the current dimension settings as a style; useful for saving overrides as a style. Prompts you: Save style: ? to list style names/Name for new dimension style - specifies the name of the style.
STatus	Displays the current value of every dimension variable.
Variables	Lists the values of dimension variables belonging to the selected dimension. Prompts you: '? to list styles/ENTER to select dimension/<Dimension style>': ? to list styles - lists the names of styles currently loaded in the drawing. ENTER to select dimension - prompts you to chose a dimension to use as the current style. Dimension style - lists the values of the style whose name you supply, as well as override values.
Restore	Resets the values of dimension variables to those of the selected dimension. Prompts you '? to list styles/ENTER to select dimension/<Dimension style>': ? to list styles - lists the names of styles currently loaded in the drawing. ENTER to select dimension - prompts you to choose a dimension whose values you want to use as the current style. Dimension style - lists the values of the style whose name you supply.

Status Bar Menu

Right-click Standard (or other dimension style name) on the status bar for the shortcut menu:



Shortcut Menu	Description
Standard	Chooses the default dimension style. When the drawing contains additional dimension styles, their names are listed here.
Properties	Opens the Dimension Styles section of the Drawing Explorer. See DimStyle command.

Procedures

To apply the current dimension style
To save a dimension style
To restore a dimension style
To list the settings in the current dimension style

Related Commands

AiDimStyle - creates Dimension Styles based on the properties of selected Dimension, Tolerance or Leader entities.
Dimstyle - creates and modifies dimension styles, through the Drawing Explorer.

DimStyleSet

Reports the current dimension style in the command bar.

Accessing the Command

command bar: *dimstyleset*

: ***dimstyleset***
Dimension style set to <current dimension style>.

The name of the current dimension style also displays in the Current Dimension Style field in the Status bar.

Command Options

No options

Related Commands

DimStyle - creates and modifies dimension styles through the Drawing Explorer.
-DimStyle - Creates and modifies dimensions styles at the command line.

DimTEdit

Changes the position of dimension text.

Accessing the Command

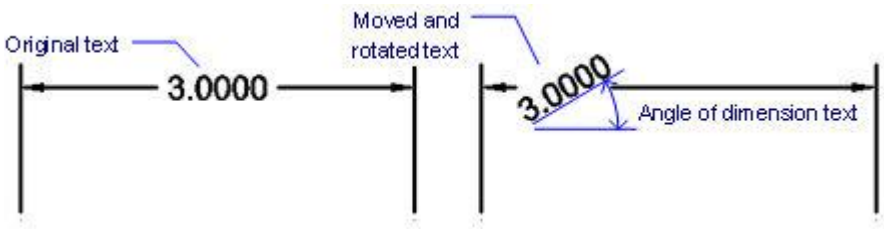
command bar: *dimtedit*
alias: *dimted*
menu bar: *Dimension | Reposition Dimension Text*
toolbar: *Dimensions |*

: ***dimtedit***

Prompts you in the command bar:

Select dimension to reposition text: (Choose a dimension.)

Specify or enter text location [Angle/Left/Center/Right/REstore]: (Drag the text to its new location, or enter an option.)



Command Options

Option	Description
REstore ( )	Resets the rotation angle to 0 degrees; does not change repositioned text.
Angle ( )	Rotates the dimension text. Prompts you: **Angle of dimension text** - specify the angle of the text; measured counter-clockwise from the positive x axis.
Left	Moves the text next to the left extension line.
Center	Centers the text between the two extension lines.
Right	Moves the text next to the left extension line.

Procedures

Editing the dimension text
To rotate the dimension text
To reposition the dimension text
To restore the text position

Related Commands

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimLeader and Leader- places leaders with text, blocks, entities, or tolerances.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

Dimensional Constraints

Applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Accessing the Command

command bar: *dimconstraint*
menu bar: *Parametric | 2D Constraints | item*
toolbar: *2D Constraints |*

: ***dimconstraint***
Prompts you in the command bar:

Select associative dimension to convert or [LInear / Horizontal / Vertical / Aligned / ANgular / Radial / Diameter] <Aligned>:

(*Select an associative dimension or choose an option.*)

Command Options

Option	Description
Select associative dimension to convert	Select an associative dimension. The dimension is converted to the dimensional constraint of the same type. This option is equivalent to the DcConvert 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 ( ).
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 ( ).
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 ( ).
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 ( ).
Radius	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 ( ).

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DimConstraint

Applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Accessing the Command

command bar: *dimconstraint*
menu bar: *Parametric | 2D Constraints | item*
toolbar: *2D Constraints |*

: ***dimconstraint***
Prompts you in the command bar:

Select associative dimension to convert or [LInear / Horizontal / Vertical / Aligned / ANgular / Radial / Diameter] <Aligned>:

(*Select an associative dimension or choose an option.*)

Command Options

Option	Description
Select associative dimension to convert	Select an associative dimension. The dimension is converted to the dimensional constraint of the same type. This option is equivalent to the DcConvert 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 ( ).
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 ( ).
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 ( ).
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 ( ).
Radius	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 ( ).

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcAligned

Constrains the distance between two points or the length of a line or polyline segment.
This command is equivalent to the Aligned option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dcaligned*
menu bar: *Parametric | 2D Constraints | Aligned*
toolbar: *2D Constraints |*

: ***dcaligned***
Prompts you in the command bar:
Specify first constraint point or [Entity/Point/2Lines] <Entity>: (*Select a valid constraint point or choose an option.*)

Specify second constraint point: (Select a valid *constraint point*.)

Specify dimension line location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)

The distance between the selected points is constrained.

Command Options

Option	Description
Entity	Constraints the length of a line, a polyline segment or the chord of an arc. Prompts you: Select an entity - Select a line, polyline segment or arc. Specify dimension location - pick a point. **Dimension text <xx.xx>** - type a value or press Enter to accept the current value
Point	Constraints the perpendicular distance between a point and a line or polyline segme Prompts you: Specify constraint point or [Line] <Line> - select a valid constraint point or choose Select line - select a line or polyline segment. Specify dimension location - pick a point. **Dimension text <xx.xx>** - type a value or press Enter to accept the current value
2Lines	Constraints the distance between 2 lines or straight polyline segments. If the entitie Prompts you: Select first line - select a line or polyline segment. Select second line to make parallel - select a line or polyline segment. Specify dimension location - pick a point. **Dimension text <xx.xx>** - type a value or press Enter to accept the current value

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcAngular

Constrains the angle between two lines or straight polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
This command is equivalent to the Angular option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dcangular*
menu bar: *Parametric | 2D Constraints | Angular*
toolbar: *2D Constraints |*

: ***dcangular***
Prompts you in the command bar:

Select first line or arc or [3Point] <3Point>: (Select a line or straight polyline segment; or an arc or arc polyline segment; or press Enter to select the 3Point option.)

If a line or straight polyline segment is selected:

Select second line: Select a line or straight polyline segment.

Specify dimension line location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)

If an arc or arc polyline segment is selected:

Specify dimension line location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type the dimension text or press Enter to accept the current dimension text.)

Command Options

Option	Description
3Point	Creates an angular constraint between three valid constraint points. Prompts you: **Specify angle vertex** - pick a point. **Specify first angle constraint point** - pick a point. **Specify second angle constraint poin**t - pick a point.

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcConvert

Converts an associative dimension to a dimensional constraint.

Accessing the Command

command bar: *dcconvert*
menu bar: *Parametric | 2D Constraints | Convert*
toolbar: *2D Constraints |*

: ***dcconvert***
Prompts you in the command bar:
Select associative dimension to convert: *(select the associative dimension you want to convert to a dimensional constraint)*

Command Options

No options.

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcDiameter

Constrains the diameter of a circle, an arc or an arc polyline segment.
This command is equivalent to the Diameter option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dcdiameter*
menu bar: *Parametric | 2D Constraints | Diameter*
toolbar: *2D Constraints |*

: ***dcdiameter***
Prompts you in the command bar:
Select arc, circle or curved polyline segment: (*Select a circle, arc or arc polyline segment.*)

Specify dimension location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)

Command Options

No options.

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcDisplay

Allows to set dimensional constraints to be displayed or hidden.
Dimensional constraints are always hidden when opening a drawing.

Accessing the Command

command bar: *dcdisplay*
menu bar: *Parametric | Show/Hide Dimensional Constraint*
toolbar: *Dimensional Constraint |*

: ***dcdisplay***
Prompts you in the command bar:

Select entities: (Select one or more entities.)
Entities in set: 1

Select entities: (Select more entities or press Enter to stop selecting.)

Enter an option [Show/Hide]<Show>: (Enter an option.)

Command Options

Option	Description
Show	Dimensional constraints on the selected entities become visible.
Hide	Dimensional constraints on the selected entities no longer display.

Procedures

To set the display of dimensional constraints of a selection of entities
To set the visibility of the dimensional constraints of all entities

Related Commands

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

DcHorizontal

Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
This command is equivalent to the Horizontal option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dchorizontal*
menu bar: *Parametric | 2D Constraints | Horizontal*
toolbar: *2D Constraints |*

: ***dchorizontal***
Prompts you in the command bar:

Specify first constraint point or [Entity] <Entity>: (Select a valid *constraint point* or press Enter to select an Entity.)

Specify second constraint point: (Select a valid *constraint point*.)

Specify dimension location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)
The horizontal distance (parallel to the X-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description
Entity	Constrains the horizontal distance between the endpoints of an entity. Prompts you: **Select an entity** - select a line, arc or polyline segment. **Specify dimension line location** - pick a point. **Dimension text <xx.xx>** - type a value or press Enter to accept the current value.

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcLinear

Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
This command is equivalent to the Linear option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dclinear*
menu bar: *Parametric | 2D Constraints | Linear*
toolbar: *2D Constraints |*

: ***dclinear***
Prompts you in the command bar:

Specify first constraint point or [Entity] <Entity>: (Select a valid *constraint point* or press Enter to select an Entity.)

Specify second constraint point: (Select a valid *constraint point*.)

Specify dimension line location: (Depending on the position of the cursor with respect to the second point, a horizontal or vertical dimension displays. Select a point to position the dimension line.)

Dimension text <current>: (Type the dimension text or press Enter to accept the current dimension text.)
The horizontal or vertical distance (parallel to the X-axis or the Y-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description
Entity	Constrains the horizontal or vertical distance between the endpoints of an entity. Prompts you: **S**elect an entity - select a line, arc or polyline segment. **Specify dimension line location** - depending on the position of the cursor a horiz **Dimension text <xx.xx>** - type a value or press Enter to accept the current valu

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcRadius

Constrains the radius of a circle, an arc or an arc polyline segment.
This command is equivalent to the Radius option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dcradius*
menu bar: *Parametric | 2D Constraints | Radius*
toolbar: *2D Constraints |*

: ***dcradius***
Prompts you in the command bar:

Select arc, circle or curved polyline segment: (Select a circle, arc or arc polyline segment.)

Specify dimension location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)

Command Options

No options.

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DcVertical

Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.
This command is equivalent to the Vertical option of the DIMCONSTRAINT command.

Accessing the Command

command bar: *dcvertical*
menu bar: *Parametric | 2D Constraints | Vertical*
toolbar: *2D Constraints |*

: ***dcvertical***

Specify first constraint point or [Entity] <Entity>: (Select a valid *constraint point* or press Enter to select an Entity.)

Specify second constraint point: (Select a valid *constraint point*.)

Specify dimension location: (Select a point to position the dimension line.)

Dimension text <xx.xx>: (Type a value or press Enter to accept the current value.)
The vertical distance (parallel to the Y-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description
Entity	Constrains the vertical distance between the endpoints of an entity. Prompts you: **Select an entity** - select a line, arc or polyline segment. **Specify dimension line location** - pick a point. **Dimension text <xx.xx>** - type a value or press Enter to accept the cu

Procedures

To edit dimensional constraints
Using expressions to define dimensional constraints

Related Commands

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.
Delconstraint - removes all dimensional and geometrical constraints from an entity.
CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.

DelConstraint

Removes all dimensional and geometrical constraints from an entity.

Accessing the Command

command bar: *delconstraint*
menu bar: *Parametric | Delete 2D Constraints*
toolbar: *2D Constraints |* ✕

: ***delconstraint***
Prompts you in the command bar:

Select entities to delete all constraints: (select an entity or entities)
Entities in set: 1

Select entities to delete all constraints: (select more entities or press Enter to delete all constraints from the selection set)

The dimensional and geometric constraints of the selected entities are deleted.

Command Options

No options

Related Commands

-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.

Direct Modeling

3dCompare

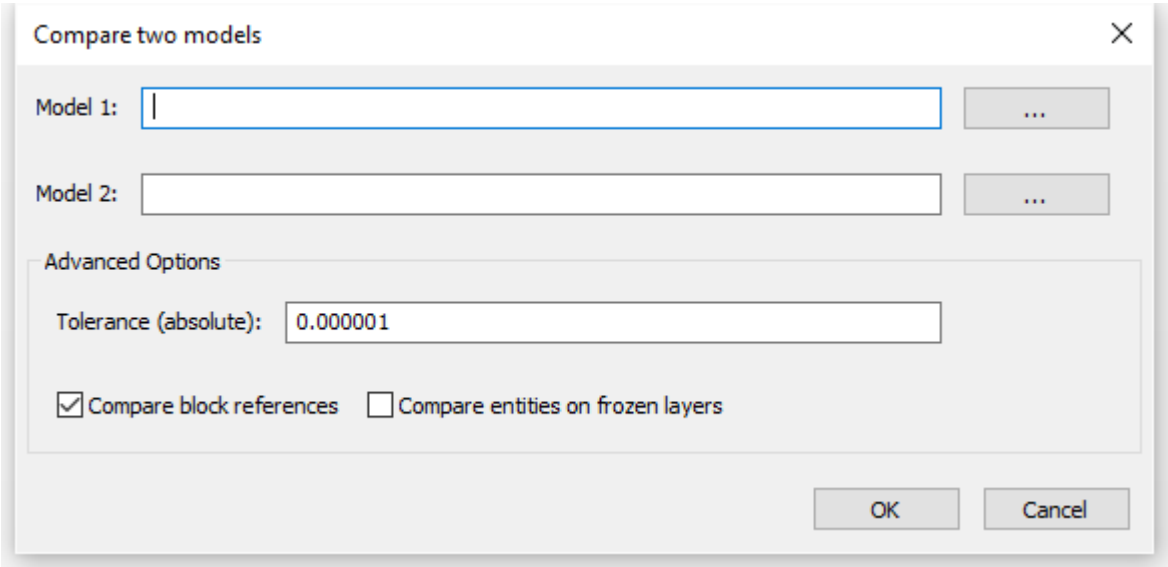
Executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel.

Accessing the Command

command bar: *3dcompare*
toolbar: *3D Compare* |

ribbon: *tools* | *Compare 3D* |

: ***3dcompare***
Displays a dialog box:



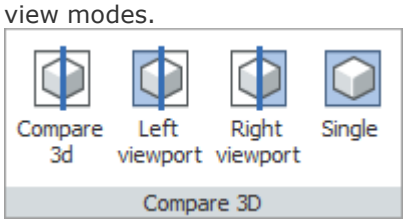
Do the following:

- Click the *Browse* button (...) of the *Model 1* and *Model 2* fields and select the two models you want to compare.
- Set the options.
- Click the *OK* button.

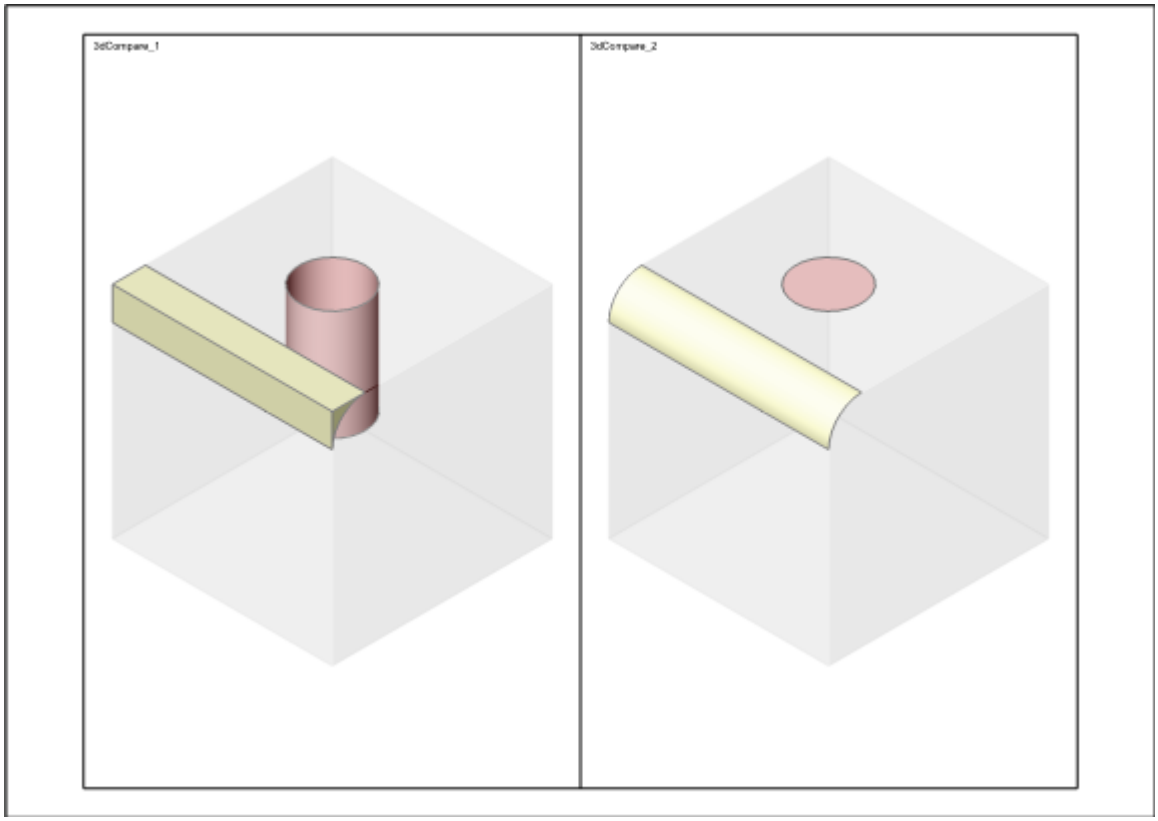
The selected files are attached to a new drawing named: Compare_<File_1>_<File_2>.dwg.

A layout named *Comparison* in this drawing contains two viewports, one for each model.

Differences between the models are reported in the Mechanical Browser panel.
The value of the 3DCOMPAREMODE system variable controls the display in the viewports of the *Comparison* layout.
The tools in the *Compare 3D* Ribbon panel of the *Tools* tab allow to switch between the various view modes.



Icon	Name	Description
	Left Viewport	Shows the differences in the left viewport of the *Comparison* layout.
	Right Viewport	Shows the differences in the right viewport of the *Comparison* layout.
	Single	Shows the differences in model space. When switching to this option, disable both layout viewport display options.



In the image above the differences display in both layout viewports.

Command Options

Option	Description
Tolerance	Threshold value for geometry comparison.
Compare block references	Specifies if the content of blocks should be compared. 3DCOMPARE can work with block structures of arbitrary depth which may contain solids in its nodes. Assemblies based on a block structure as well as regular blocks and inserts can be analyzed also.
Compare entities on frozen layers	Specifies if entities on frozen layers should be compared.

Related Commands

BmBrowser - toggles the visibility of the Mechanical Browser panel.
DwgCompare - compares a selected drawing with the current drawing during a comparison session.

DmAngle3d

Applies an angle constraint between faces and/or edges of 3D solids. Allows to control the top angle of a cone.

Accessing the Command

command bar: *dmangle3d*
menu bar: *Parametric | 3D Constraints | Angle*
toolbar: *3D Constraints* | or
Quad: *Constraints* | or

: ***dmAngle3d***
Prompts you in the command bar:

Select first entity or specify [cone Angle constraint/World/Block]: (Select a face or an edge.)
Entities/subentities in set: 1

Select second entity: (Select a face or an edge.)

Specify angle value or set [Axis] <90.00>: (Either press Enter to accept the current value, or type a new value.)

Press the TAB key to select obscured geometry.

Command Options

Option	Description
Cone angle constraint	Allows to control the top angle of a cone by specifying angle between the axis and the cone face (= cone half angle); prompts you: Select a cone face: Select the face of a cone. You are prompted: Specify angle value <xx.xx>: Type a value. Values must be greater than 0°and smaller dan 90°.
Axis	Allows to specify the rotation axis; prompts you: Select single entity/subentity: Select a 3D solid edge or a line. You are prompted: Specify angle value or [Change axis/Remove axis] <xx.xx>: Do one of the following: • Type a value to specify the angle. • Choose *Change axis* to select a different axis. • Choose *Remove axis* to remove the previously selected axis.
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmAudit

Fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DmAudit is a powerful tool to analyze and automatically fix problems in 3D geometry, which are caused by incorrect operations or if geometry was created in another CAD package and imported using the Import command. It is recommended to run DmAudit whenever 3D geometry is imported, if the Import command is not set up to do this automatically. After fixing 3D geometry using DmAudit, operations such as Boolean operations, direct modeling and generating drawing views are possible.
The DMAUDITLEVEL system variable controls whether certain types of errors are reported by DmAudit or not. If set to 1 then modelling kernel dynamic range errors are ignored.
If DMREPORTPANEL= ON, BricsCAD reports the results in the Mechanical Browser / Report panel instead of the command bar.

Accessing the Command

command bar: *dmaudit*

: ***dmaudit***
Prompts you in the command bar:

Select entities to audit [Entire model] <Entire model>: (Select an entity to include in the operation or press Enter to analyze the entire model.)

Entities in set: 1

Select entities to audit [Entire model] <Entire model>: *(Select more entities or press Enter.)*

Entities in set: xx

Choose action [Check/Fix/Options] <Fix>: (Choose an option or press Enter.)

Command Options

Option	Description
Check	Checks the 3D geometry in the drawing for errors without attempting to fix them. Can be used to check whether a model has problems which prevent processing.
Fix	Checks the 3D geometry in the drawing for errors and attempts to fix them.

Options	Prompts you:
	Select option [Back/report Panel/copy to File/select error Level] <Back>:
	• Back: Returns to the main prompt of the command.
	• Report Panel: Redirects the output of the command to the Report panel.
	• Copy to File: Reports the output of the command in a file.
	• Select Error Level: Allows to set the strictness of the check. Three levels are available:
	  • Critical: Thorough check of the model.
	  • General: If no errors of this level are found or fixed, modeling operations will succeed.
	  • Warnings: Reports minor problems, which do not prevent modeling operations but might slow down the overall performance.

Related Commands

dmSimplify - simplifies the geometry and topology of 3d solid entities.

dmStitch - converts a set of region and surface entities that bound a watertight area to a 3d solid.
Import - imports the geometry from external files into the current drawing.

DmChamfer

Creates an equal distance chamfer between adjacent faces sharing a sharp edge.

Accessing the Command

command bar: *dmchamfer*
menu bar: *Model | Direct Modeling | Chamfer*
toolbar: *Direct Modeling |*

: ***dmchamfer***
Prompts you in the command bar:
Select edges to create chamfer: *(Select the edges you want to chamfer. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)*

The chamfer is applied dynamically. To define the chamfer distance, do one of the following:

- Click the left mouse button.
- Type a value in the dynamic input field.

Command Options

No options

Related Commands

dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmMove - moves the selected faces or edges of a solid using a vector.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmRotate - rotates faces of a solid around an axis.

Chamfer - creates chamfers; adds an angle to intersections.
Fillet - fillets intersections; joins intersecting lines with an arc of any suitable radius.

DmCoincident3d

Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.

Accessing the Command

command bar: *dmcoincident3d*
menu bar: *Parametric | 3D Constraints | Coincident*
toolbar: *3D Constraints |*
Quad: *Constraints | or*

: ***dmCoincident3d***
Prompts you in the command bar:

Select a pair of subentities or [Autoconstrain/WOrld/BLock] <Autoconstrain>: (Select the first edge or face or press Enter.)

Select second subentity [WOrld/BLock]: (Select the second edge or face.)

Press the TAB key to select obscured geometry.

Command Options

Option	Description
Autoconstrain	Automatically creates a coincident constraint between each pair of connected faces in selection of solids. Prompts you: Select a set of entities: Select at least two solids. Entities in set: 2 Select a set of entities: Select more solids or press Enter. You can check which constraints are created in the Mechanical Browser panel.
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - forces 3d constraints to update.
BmBrowser - toggles the visibility of the Mechanical Browser window.

DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmConcentric3d

Applies a concentric constraint between two cylindrical, spherical or conical surfaces.

Accessing the Command

command bar: *dmconcentric3d*
menu bar: *Parametric | 3D Constraints | Concentric*
toolbar: *3D Constraints |*

: ***dmConcentric3d***
Prompts you in the command bar:
Select first subentity [WOrld/BLock]: *(Select the first cylindrical (circular or elliptical), spherical or conical surface or choose an option.)*
Select second subentity [WOrld/BLock]: *(Select the second cylindrical (circular or elliptical), spherical or conical surface or choose an option.)*

Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: **Select block reference** - Select a block reference of which the reference coordinate system will be used as an argument. **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] -** Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmConstraint3d

Applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.

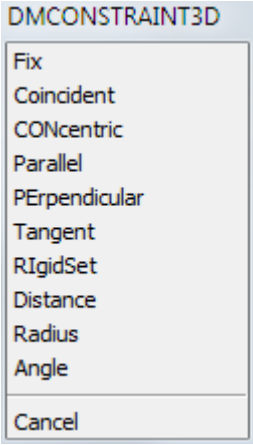
Accessing the Command

command bar: *DmConstraint3d*
menu bar: *Parametric | 3D Constraints | item*
toolbar: *3D Constraints |*

: ***dmconstraint3d***
Prompts you in the command bar:

Select type of 3D constraint [Fix/Coincident/CONcentric/Parallel/PErpendicular/Tangent/RIgidSet/Distance/Radius/Angle]: (Type a 3D constraint type.)

Displays a prompt menu:



(Choose a 3D constraint in the prompt menu.)

Command Options

Option	Description
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 ( ).
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 ( ).
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 ( ).
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 to faces of a solid or of different solids. The option is equivalent to the DmAngle3d command ( ).
Rigid Set	Allows to define a set of entities or subentities as a rigid body. The option is equivalent to the DmRigidSet3d command ( ).

Related Commands

BmBrowser - displays the Mechanical Browser
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmCopyFaces

Copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.

Accessing the Command

command bar: *dmcopyfaces*
menu bar: *Model | Direct Modeling | Copy Faces*
toolbar: *Direct Modeling |*
ribbon: *Solid | Edit | Copy Faces*

: ***dmcopyfaces***
Prompts you in the command bar:

Select several entities/subentites: (Select one or more entities.)

Entities in set: 1

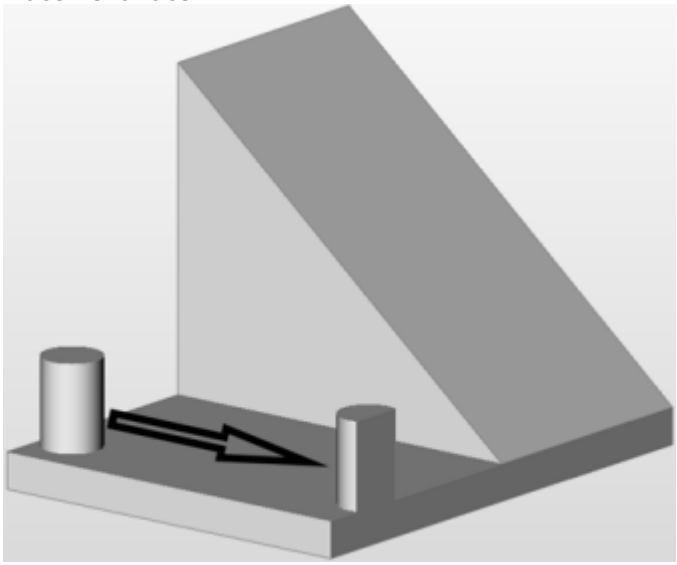
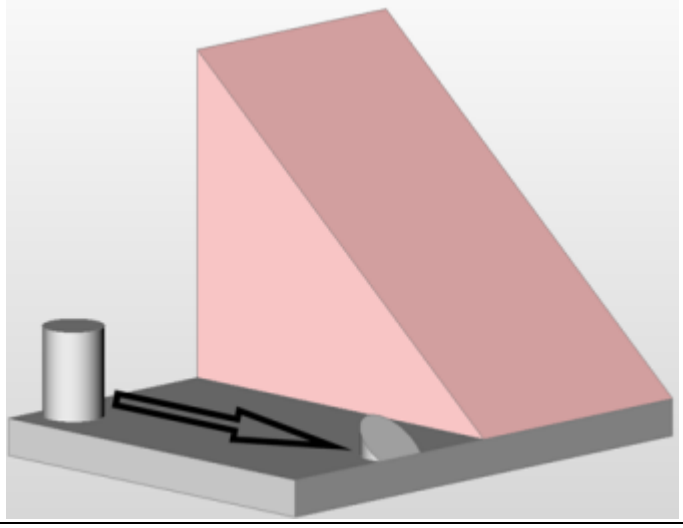
Select several entities/subentites: (Select more entities or, press Enter stop selecting entities.)

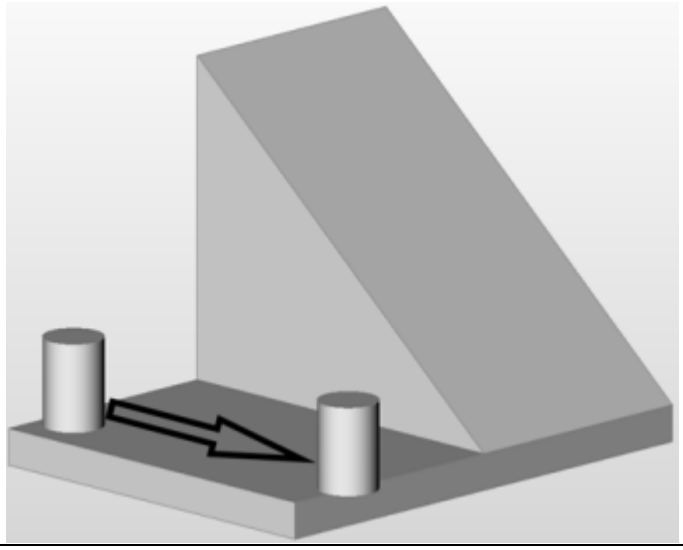
Enter base point: (Pick a point.)

Select insertion point or [Rotate/Multiple/Mode]: (Pick another point, or enter an option.)

Command Options

Option	Description
Rotate	Allows to rotate the selected entities; prompts you: Specify rotation angle - type a value Select insertion point or [Rotate/mUltiple/MOde] - pick a point or choose an option.
Multiple	Allows to create multiple copies: prompts you: Select insertion point or [Rotate/mUltiple/MOde] - pick a point or choose an option for the first copy. Select insertion point or [Rotate/mUltiple/MOde] - pick a point or choose an option for the next copy. Select insertion point or [Rotate/mUltiple/MOde] - pick a point or choose an option for the next copy or press Enter to stop.
Mode	Toggles between various cutting options; prompts you: Cutting mode [Placement face/Select faces/No cut] <Placement face> - choose an option.

	• Placement Face:  • Select Faces: 

	• No Cut: 

Procedures

Copying features

Related Commands

BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.

DmDeformCurve

Deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

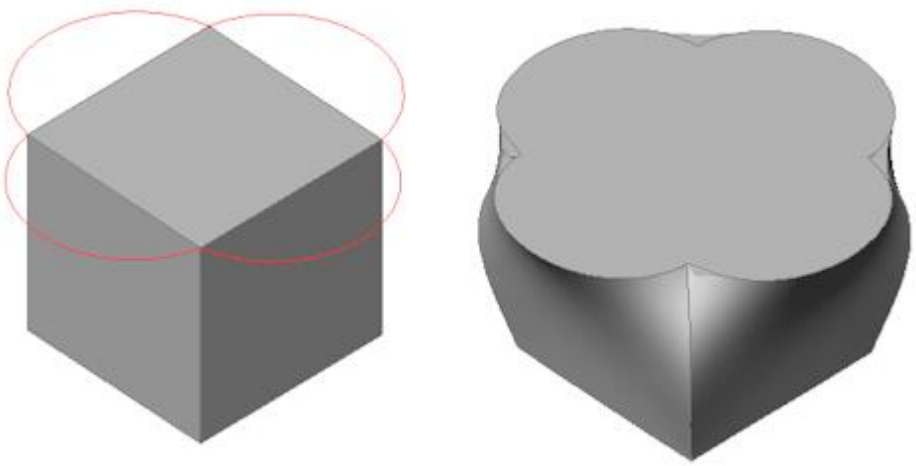
Accessing the Command

command bar: *dmdeformcurve*
menu bar: *Model | Direct Modeling | Deform | Transform Curve*
toolbar: *Direct Modeling |*

: ***dmdeformcurve***
Prompts you in the command bar:
Select edge to deform or [add Faces] <done>: (Select an edge.)

Select target curve: (Select a curved to replace the selected edge with.)

Select edge to deform or [add Faces] <done>: (Select an edge or press Enter to conclude the command.)



Command Options

Option	Description
Add Faces	Allows to select additional faces to deform (in addition to the faces adjacent to selected edges). Prompts you: **Select additional deformable faces:** Select a face. Entities in set: 1 **Select additional deformable faces:** Select a face or press Enter to stop selecting additional faces.

Related Commands

dmDeformMove - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.
dmDeformPoint - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.
dmTwist - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

DmDeformMove

Deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

Accessing the Command

command bar: *dmdeformmove*
menu bar: *Model | Direct Modeling | Deform | Move Edge*
toolbar: *Direct Modeling |*

: ***dmdeformmove***
Prompts you in the command bar:
Select edges of a 3D solid, surface or region: (Select an edge.)

Entities in set: 1

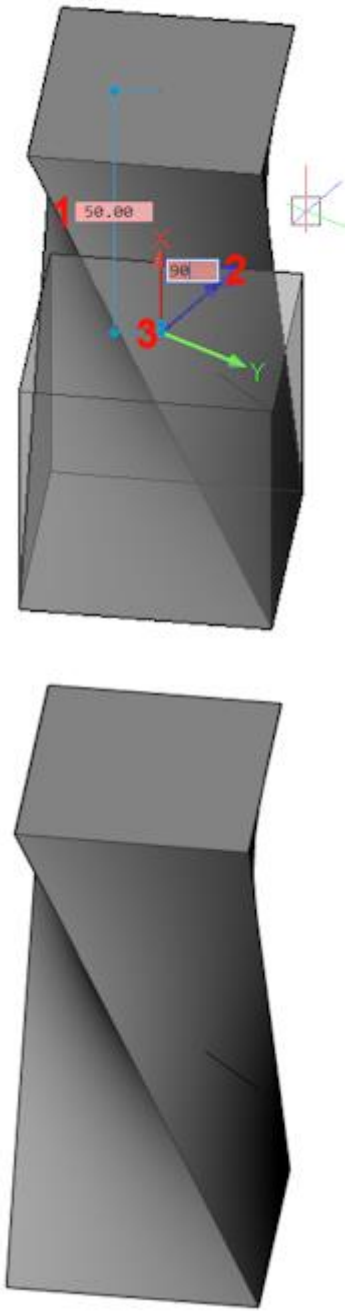
Select edges of a 3D solid, surface or region: (Select more edges or press Enter.)

The selected edges are moved dynamically forcing the deformation of the adjacent faces.

Specify deformation value or set [Base point/Direction/additional Faces]: (Specify the deformation or choose an option.)

Command Options

Option	Description
Specify Deformation Value	To specify the deformation do one of the following: • Specify a point in the drawing. • Type a value in the dynamic entry fields. Hit the TAB key to switch between the distance (1) and angle (2) fields. The edges are rotated around an axis passing through the base point (3) in the deformation direction.

	 *Dynamics (left) and final result (right)*
Base Pöint	Allows to specify the base point. The deformation axis passes through the base point. Prompts you: **Specify new deformation point:** Specify a point in the drawing.

Direction	Allows to specify the direction of the deformation axis. Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:** Choose an option or press Enter to define the direction by specifying two points in the drawing.
Additional Faces	Allows to select additional faces to be deformed, in addition to ones adjacent to the selected edge(s). Prompts you: Select additional deformable faces: Select a face. Entities in set: 1 Select additional deformable faces: Select more faces or press Enter.

Related Commands

dmDeformCurve - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.
dmDeformPoint - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.
dmTwist - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

DmDeformPoint

Deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. The selected faces are deformed as smooth as possible. If there is G1 or G2 continuity between faces, it is preserved after deformation.

Accessing the Command

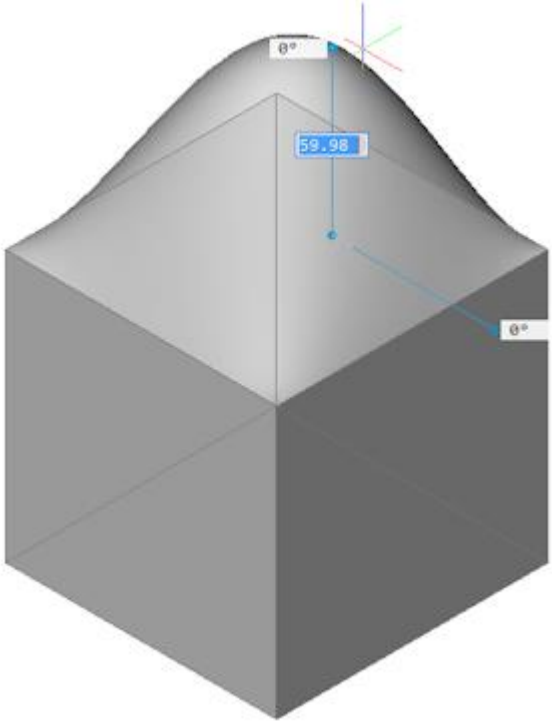
command bar: *dmdeformpoint*
menu bar: *Model | Direct Modeling | Deform | Move Point*
toolbar: *Direct Modeling |*

: ***dmdeformpoint***
Prompts you in the command bar:

Select faces of a 3D solid or surface: (Select a face of a 3D solid or a surface or a region.)
Entities in set: 1

Select faces of a 3D solid or surface: (Select a connected face of the same 3D solid or press Enter.)
Specify deformation point or [change Parameters] <use default point>: (Specify a point or press Enter to accept the default point.)
The selected entities are deformed dynamically towards the surface normal direction at the deformation point.

Specify deformation value or [set Target point/set Direction/change Base point]: (Specify a point or type a value.)



Command Options

Option	Description

Change Parameters	Prompts you:
	Change parameter [Alpha/Beta/Gamma/Delta] <done>:
	• Alpha: Specifies the resistance to stretching. It is a second order tensor which can be described with three numbers. Each value must be either 0 or positive number:
	  • alpha u: u-direction resistance
	  • alpha v: v-direction resistance,
	  • alpha theta: the angle between the 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 positive number.
	• Gamma: Specifies the resistance to the rate of change in bending. The value must be either 0 or positive number.
	• Delta: Specifies the resistance to deviations from the default shape. The value must be either 0 or positive number.

Set Target Point	Prompts you: **Specify deformation target point:** Specify a point. The entities are deformed towards the specified point.
Set Direction	Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:** Choose an option or press Enter to define the extrusion direction by specifying two points. The selected entities are deformed dynamically in the specified direction. Prompts you: **Specify deformation value or [set Target point/set Direction/change Base point]:** Specify a point or type a value or choose an option.
Change Base Point	Prompts you: Specify new deformation point: Specify a point. Prompts you: **Specify deformation value or [set Target point/set Direction/change Base point]:** Specify a point or type a value or choose an option.

Related Commands

dmDeformCurve - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.
dmDeformMove - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.
dmTwist - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

DmDelete

The command is obsolete. The functionalities of DmDelete are integrated in the Erase command.

DmDistance3d

Applies a distance constraint between two subentities of a solid or of different solids.

Accessing the Command

command bar: *dmdistance3d*
menu bar: *Parametric | 3D Constraints | Distance*
toolbar: *3D Constraints |*
Quad: Constraints |

: ***dmDistance3d***
Select a first entity or specify [measurement Mode/WOrld/BLock]: (Select the first edge, face or surface or choose a measurement mode.)
Select a first entity or specify [measurement Mode]: *(Select the first edge, face or surface or choose a measurement mode.)*
Press the TAB key to select obscured geometry.
Enter distance value <xxx.xx>: (Type the desired distance.)

Command Options

Option	Description
Measurement Mode	Allows to specify how the distance must be measured between cylindrical and spherical faces. The default measurement mode is *Axis*. Prompts you: Specify measurement mode [Axis/Central point/Boundary] <Axis>: Choose a measurement. Prompts you: Select a second entity or specify [measurement Mode]: Select the second edge, face or surface or choose a measurement mode for the second entity. Prompts you: Enter distance value <xxx.xx>: *(Type the desired distance.)*
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmExtrude

Creates 3D solids or 3D surfaces by extruding closed 2D entities, faces of 3D solids, regions or closed boundaries.
Optionally adds volume to solids or removes volume from solids.

Accessing the Command

command bar: *dmextrude*
menu bar: *Model | Direct Modeling | Extrude*
toolbar: *Direct Modeling |* *(solid)* *(surface)*
Quad cursor menu: *Direct Modeling |* *(solid)* *(surface)*

: ***dmExtrude***
When launched from the command bar, menu bar, toolbar or ribbon:

Prompts you in the command bar:

Select entities/subentities to extrude or set [MOde]: (Select a region or a closed 2D entity (closed polylines, circles and ellipses are accepted) or a face of solid.)

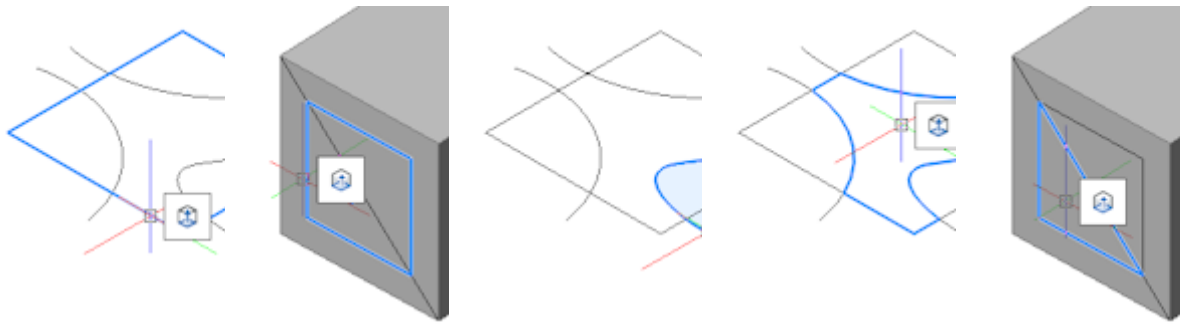
Entities in set: 1

Select entities/subentities to extrude or set [MOde]: (Select another entity or subentity or press Enter to stop selecting entities.)

When launched from the Quad cursor menu:

First select the entity, entities or boundary to be extruded.

Closed 2D entities (1)(2) and regions (3) are accepted. Closed boundaries in the XY-plane of the current coordinate system (4) or on the face of a solid (5) are recognized automatically if the *Select detected boundaries* option ( ) on the *Selection Modes* toolbar is active.



1 2 3 4 5

Do one of the following:

- Launch the command when the desired entity or boundary highlights.
- To extrude multiple entities: compose a selection set, then launch the command. Click to add the highlighted entity to the selection.

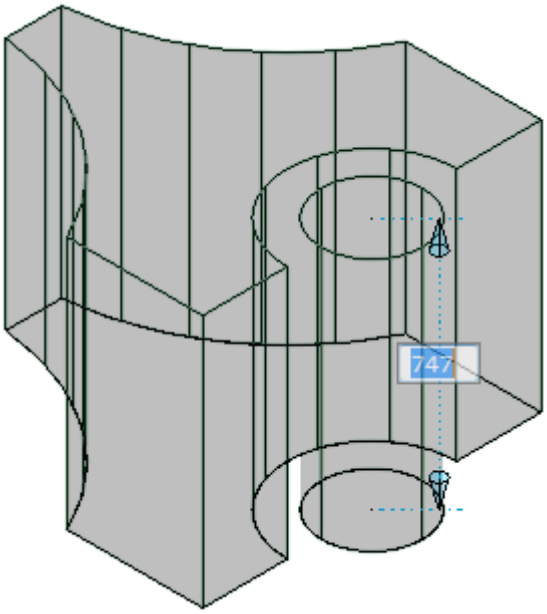
Solid mode:

Specify height of extrusion or [Auto(subtract or create)/Create/SUbtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>:

The selected entities are extruded dynamically. *(Move the mouse to specify the extrusion direction.)*
The extrusion direction is perpendicular to the selected entity or entities.

Do one of the following to define the height of the extrusion:

- Specify a point in the drawing.
- Type a value in the dynamic entry field.
- Choose the Limit option.



Surface mode:

Specify height of extrusion or set [Auto/Create/Slice/Both sides/Taper angle/Direction] <Auto>:

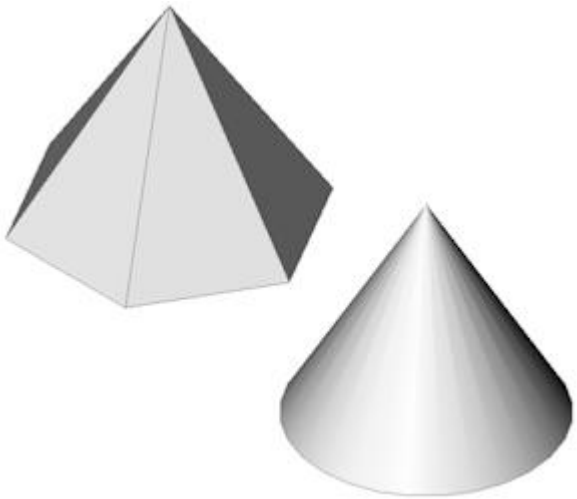
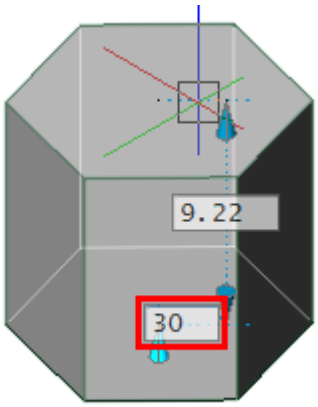
The selected entities are extruded dynamically. Move the mouse to specify the extrusion direction. The extrusion direction is perpendicular to the selected entity or entities, unless the *Direction* option is selected.

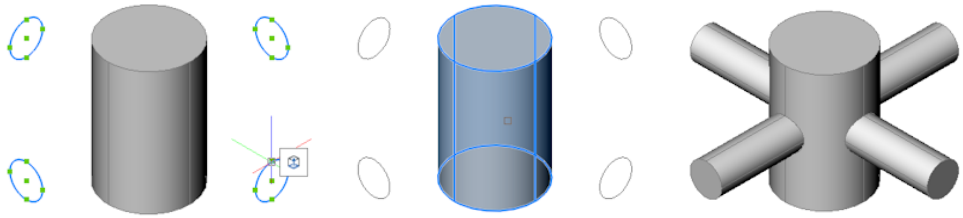
Do one of the following to define the height of the extrusion:

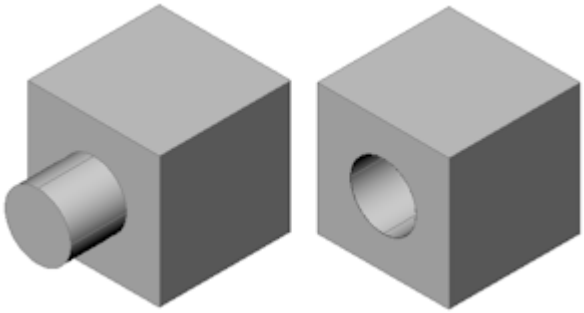
- Specify a point in the drawing.
- Type a value in the dynamic entry field.

Command Options

Option	Description
Mode	Allows to create either solids or surfaces. Prompts you: **Choose type of created entity [SOlid/SUrface] <Solid>:** • **Solid**: creates solids. • **Surface**: creates surfaces. The *Extrude* tool ( ) starts the command in *Solid* mode. The *Extrude Surface* tool ( ) starts the command in *Surface* mode.
Select Entities	Click an entity or choose an option in the dmExtrude prompt menu. Depending on the value of the DELOBJ (Delete Object) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities.

Taper Angle	Specifies the taper angle for the extrusion. Prompts you: **Specify taper angle value in degrees -** type a value in the command bar. The angle is measured from the extrusion direction. A negative value tapers outwards. This option allows to create (truncated) pyramids and cones.  If *Dynamic Dimensions* are ON, press the TAB key to jump to the *Taper Angle* field, then type the desired value and press the TAB key again to apply the new value. 
Direction	Allows to specify the extrusion direction. Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:** Select an option: • 2 Points: Define the extrusion direction by specifying two points. • X axis: Sets the extrusion direction parallel to the X axis of the current UCS. • Y axis: Sets the extrusion direction parallel to the Y axis of the current UCS. • Z axis: Sets the extrusion direction parallel to the Z axis of the current UCS.
Limit	Allows to limit the extrusion by the face of a solid. Prompts you:

	Select face or 3d solid to limit extrude: Move the cursor over a face, then click when the face highlights.  1. Select entities. 2. Launch DMEXTRUDE. 3. Choose Limit option. 4. Select cylindrical face. Result The SELECTIONPREVIEW system variable must be set to 2 or 3 to highlight faces. Press the TAB key to subsequently highlight overlapping faces.

Auto*	The result depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable.
	• when extruded inside the existing solid, the new solid is subtracted from the existing solid (right).
	 
	• when extruded outside the existing solid (left) the result depends on the value of the DMEXTRUDEMODE system variable, which is stored as the sum of the value of the selected options.
	  • **0**: the new solid is unified with the existing solid. There is no interaction with intersecting solids. This is the default option for 3D Modelling.
	  • **1**: a separate solid is created which is unified with intersecting solids.
	  • **2**: the new solid is unified with the existing solid and subtracted from intersecting solids.
	  • **3**: a separate solid is created and subtracted from intersecting solids. This is the default option for BIM.

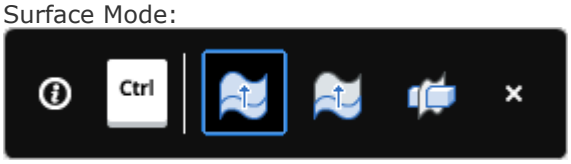
Create*	Regardless of the extrusion direction, a new 3D solid is created.
Subtract *	The 3D solid is subtracted from each interfering existing solid.
Unite*	The new 3D solid is unified with each interfering existing solid.
Slice*	The extruded surface cuts through solids, much like the Slice command.

Both Sides	Extrudes the selected boundary (solid mode) or linear entities (surface mode) symmetrically at both sides.

* Repeatedly hit the Ctrl-key during the dynamic display of the extrusion to cycle the various options: Auto / Create / Subtract / Unite showing the result of the currently selected option.
If TIPS is ON the *Tips* widget displays:
Solid Mode:

- : Auto
- : Create
- : Subtract
- : Unite

Surface Mode:



- : Auto
- : Create
- : Slice

Related CommandsC

Extrude - Extrudes 2D entities to 3D solids.
Loft - creates 3D solids passing through two or more cross sections.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.

dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmMove - moves the selected faces or edges of a solid using a vector.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmRotate - rotates faces of a solid around an axis.
dmStitch - converts a set of region and surface entities that bound a watertight area to a 3d solid.
dmThicken - creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, and faces of 3D solids.

DmFillet

Creates a smooth fillet between adjacent faces sharing a sharp edge.

Accessing the Command

command bar: *dmfillet*
menu bar: *Model | Direct Modeling | Fillet*
toolbar: *Direct Modeling |*

: ***dmfillet***
Prompts you in the command bar:
Select edges to create fillet: *(Select the edges you want to fillet. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)*

The fillet is applied dynamically. To define the fillet radius, do one of the following:

- Click the left mouse button.
- Type a value in the dynamic input field.

Command Options

No options.

Related Commands

dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmMove - moves the selected faces or edges of a solid using a vector.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmRotate - rotates faces of a solid around an axis.

Chamfer - creates chamfers; adds an angle to intersections.
Fillet - fillets intersections; joins intersecting lines with an arc of any suitable radius.

DmFix3d

Applies a fixed constraint to a solid or to an edge or a face of a solid.

Accessing the Command

command bar: *dmfix3d*
menu bar: *Parametric | 3D Constraints | Fix*
toolbar: *3D Constraints |*

: ***dmFix3d***
Prompts you in the command bar:
Select an edge, face or 3D solid or [Multiple/BLock]: *(Select a solid, or an edge or a face of a solid.)*
Press the TAB key to select obscured geometry.

Command Options

Option	Description
Solid	All faces and edges of the solid are fixed.
Face	The selected face is fixed and cannot be moved or rotated.
Edge	The selected edge is fixed. The adjacent faces can be rotated, but not moved.
Multiple	Allows to select multiple entities or subentities; prompts you: Select edges, faces or 3D entities - Select a solid, or an edge or a face of a solid Entities in set: 1 Select edges, faces or 3D entities - Select more solids, or edges or faces of solids or press Enter.
Block	

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmGroup

Creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

A group feature can be coupled with its description, which can be used to store manufacturing instructions or any other informations 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.

Accessing the Command

command bar: *dmgroup*

: ***dmgroup***

Prompts you in the command bar:

Enter parameter option [New/Edit/Dissolve] <New>: (Press Enter to create a new group or choose an option.)

Enter name for new group <Group_1>: (Enter name* of the new group feature or press Enter to accept the Group_xx default name.)

Select entities: (Select faces or edges of solids.)

Entities in set: 1

Select entities: (Select more faces or edges of 3D solids or press Enter to stop selecting)

Enter description for new group: (Type a description. which appears in the Description field of the feature in Mechanical Browser or press Enter to keep the description field empty.)

Command Options

Option	Description
Edit	Allows to edit an existing group feature. Prompts you: Enter name of group * - type the name of the group to be edited. Select entities - Select faces or edges of solids. Entities in set: 1 Select entities - Select more faces or edges of 3D solids or press Enter to stop selecting. The new selection replaces the previous collection.
Dissolve	Allows to delete an existing group feature.. Prompts you: Enter name of group * - type the name of the group to be removed. The group feature is deleted.

* Group feature names are case sensitive.

Related Commands

Group - creates and modifies named groups of entities.
DmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
DmSelectEdges - selects the edges of faces and solids.
SmSelect - selects hard edges or form features.

DmMove

Moves solids, faces or edges of a solid, or inserts using a vector.

Accessing the Command

command bar: *dmmove*
menu bar: *Model | Direct Modeling | Move*
toolbar: *Direct Modeling |*

: ***dmmove***
Prompts you in the command bar:

Select several entities/subentities: (Select the geometry you want to move. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting)

Enter base point <0,0,0>: (Specify the base point for the move operation.)

The selected geometry moves dynamically. To specify the endpoint, do one of the following:

- Click the left mouse button.
- Type a value in the dynamic input field.

Command Options

No options.

Related Commands

Align - aligns one entity with another through moving, rotating, and scaling.
dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmRotate - rotates faces of a solid around an axis.

DmParallel3d

Applies a parallel constraint between two faces of a solid or of different solids.

Accessing the Command

command bar: *dmparallel3d*
menu bar: *Parametric | 3D Constraints | Parallel*
toolbar: *3D Constraints |*

: ***dmParallel3d***
Prompts you in the command bar:
Select first subentity [WOrld/BLock]: *(Select the first face or choose an option.)*
Select first subentity [WOrld/BLock]: *(Select the second face or choose an option.)*

Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: **Select block reference** - Select a block reference of which the reference coordinate system will be used as an argument. **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] -** Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmPerpendicular3d

Applies a perpendicular constraint between two faces of a solid or of different solids.

Accessing the Command

command bar: *dmperpendicular3d*
menu bar: *Parametric | 3D Constraints | Perpendicular*
toolbar: *3D Constraints |*

: dmPerpendicular3d
Prompts you in the command bar:

Select first subentity [WOrld/BLock]: *(Select the first face or choose an option.)*
Select second subentity [WOrld/BLock]: *(Select the second face or choose an option.)*

Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmPushPull

Adds or removes volume from a solid by moving a face.

Accessing the Command

command bar: *dmpushpull*
menu bar: *Model | Direct Modeling | Push/Pull*
toolbar: *Direct Modeling |*

: ***dmpushpull***
Prompts you in the command bar:

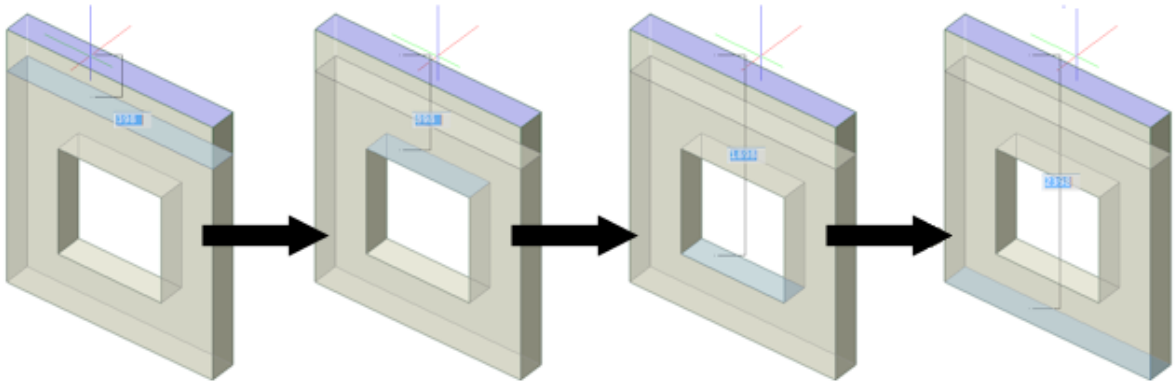
Select faces to push/pull: (select one or more faces or detected boundaries on the same or different solids; press and hold the TAB key to select hidden geometry. Press Enter to stop selecting geometry.)
Entities in set: 2

Specify distance: (Specify a point or type a distance.)

The selected geometry is edited dynamically.

Do one of the following:

- Specify a point.
- Type a value in the dynamic entry field.
  Hit the TAB key to select a different reference face. The current reference face highlights. All faces which are parallel to the face being pushpulled are recognized, the nearest first. Press and hold the Shift key when hitting the TAB key to cycle reference faces in reverse order.



Command Options

Option	Description
Planar faces	The faces are moved, thus adding or removing volume from the solid(s). The value of the movement vector displays in the dynamic entry field.
Cylindrical and spherical faces	Edits the radius. The current radius displays in the dynamic entry field.
Toroidal faces	Edits the tube radius of the torus. The current radius displays in the dynamic entry field.

Operations

To control dimensions

Related Commands

dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmMove - moves the selected faces or edges of a solid using a vector.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmRotate - rotates faces of a solid around an axis.

BimDrag - drags major faces of solids while preserving connections with minor faces. Drags minor faces; optionally connects minor faces to major faces of other solids.

DmRadius3d

Applies a radius constraint to cylindrical surfaces or circular edges.

Accessing the Command

command bar: *dmradius3d*
menu bar: *Parametric | 3D Constraints | Radius*
toolbar: *3D Constraints |*

: ***dmRadius3d***

Select a face or circular edge: (Select a cylindrical surface or a circular edge)

Enter radius value <current value>: (Either press Enter to accept the current value, or type a new value.)

Press the TAB key to select obscured geometry.

In case the surface of an elliptical cylinder is selected, the surface is converted to a circular cylinder.

Command Options

No options.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.

DmRevolve

Creates 3D solids by revolution of closed 2D entities or regions about an axis.

Accessing the Command

command bar: *dmrevolve*
menu bar: *Model | Direct Modeling | Revolve*
toolbar: *Direct Modeling |* *(solid)* *(surface)*
Quad cursor menu: *Direct Modeling |* *(solid)* *(surface)*

: ***dmRevolve***

When launched from the command bar, menu bar, toolbar or ribbon:

Prompts you in the command bar:

Select entities/subentities to revolve:

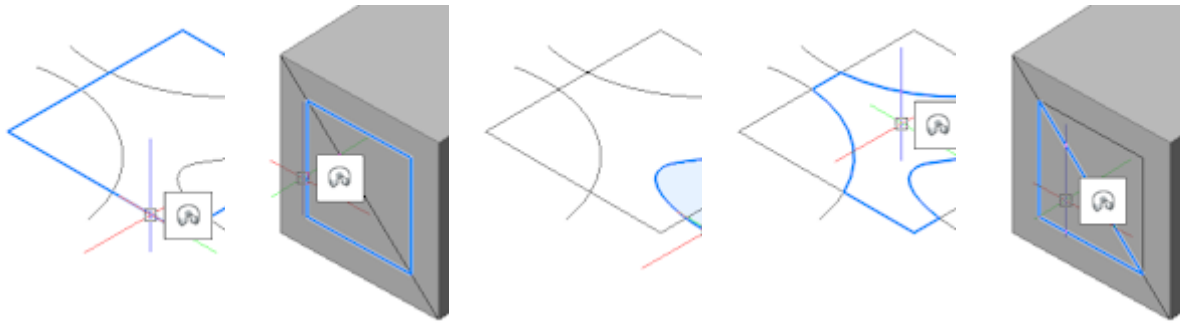
Select planar entities/subentities to revolve or set [MOde]: (Select a region or a closed 2D entity (closed polylines, circles and ellipses are accepted) or a face of solid.)
Entities/subentities in set: 1

Select planar entities/subentities to extrude or set [MOde]: (Select another entity or subentity or press Enter to stop selecting entities.)

When launched from the Quad cursor menu:

First select the entity, entities or boundary to be extruded.

Closed 2D entities (1)(2) and regions (2) are accepted. Closed boundaries in the XY-plane of the current coordinate system (4) or on the face of a solid (5) are recognized automatically if the *Select detected boundaries* option ( ) on the *Selection Modes* toolbar is active.



1 2 3 4 5

Do one of the following:

- Launch the command when the desired entity or boundary highlights.
- To revolve multiple entities: compose a selection set, then launch the command.

Solid mode:

Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: (Select a linear entity, or select an option, or right click to define the axis by two points.)
The selected entities are revolved dynamically. Move the mouse to specify the angle direction.

Specify angle of revolving or [Auto/Create/SUbtract/Unite] <Auto>: (Choose an option, then specify the angle.)

Surface mode:

Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: (Select a linear entity, or select an option, or right click to define the axis by two points.)

The selected entities are revolved dynamically. Move the mouse to specify the angle direction.

Specify angle of revolving or [Auto/Create] <Create>:

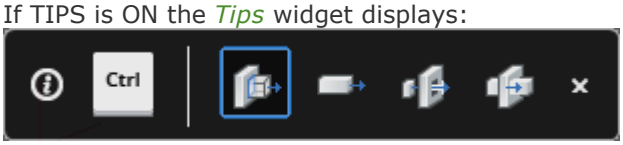
Command Options

Option	Description
Mode	Allows to create either solids or surfaces. Prompts you: **Choose type of created entity [SOlid/SUrface] <Solid>:** • Solid: creates solids. • **Surface**: creates surfaces. The *Revolve* tool ( ) starts the command in *Solid* mode. The *Revolve Surface* tool ( ) starts the command in *Surface* mode.
Select Entities/Subentities to extrude	Click an entity or choose an option in the dmRevolve prompt menu. Depending on the value of the DELOBJ (Delete Object) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities.
Select a linear (sub)entity as axis or define axis by <2Points>/Xaxis/Yaxis/Zaxis:	Select a line or an edge of a solid. Edges of a solid can only be selected if the *Edges* option of the SELECTIONMODES system variable is set. Click the *Edges* tool button ( ) on the *Selection Modes* toolbar to toggle this setting. A pressed button indicates the option is set.
2Points	Specify start point of axis <0,0,0>: Specify the start point of the axis or press Enter to accept the origin point of the current UCS. Specify endpoint of axis or <use axis parallel to view direction>: Specify the end point of the axis or press Enter to set the axis parallel to the view direction (= perpendicular to the screen).
Zaxis	Uses the x axis of the current UCS as the axis of revolution
Yaxis	Uses the y axis of the current UCS as the axis of revolution.
Xaxis	Uses the z axis of the current UCS as the axis of revolution.
Press the Ctrl key to cycle the various options.	
Auto*	The result depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable. • when revolved inside the existing solid, the new solid is subtracted from the existing solid. • when revolved outside the existing solid:

	• If DMEXTRUDEMODE = 0: the new solid is unified with the existing solid (Auto (Subtract or Unite)) • If DMEXTRUDEMODE = 1: a separate solid is created (Auto (Subtract or Create))
Create*	Regardless of the revolving direction, a new 3D solid is created.
Subtract*	The 3D solid is subtracted from each interfering existing solid.
Unite*	The new 3D solid is unified with each interfering existing solid.

* Repeatedly hit the Ctrl-key during the dynamic display of the extrusion to cycle the various options: Auto / Create / Subtract / Unite showing the result of the currently selected option.

If TIPS is ON the *Tips* widget displays:



- : Auto
- : Create
- : Subtract
- : Unite

Related Commands

Loft - creates 3D solids passing through two or more cross sections.
Revolve - revolves open or closed 2D entities and solid edges and faces to turn them into 3D solids or surfaces.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.

dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmMove - moves the selected faces or edges of a solid using a vector.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRotate - rotates faces of a solid around an axis.

DmRigidSet3d

Allows to define a set of entities or subentities as a rigid body.

Accessing the Command

command bar: *dmrigidset3d*
menu bar: *Parametric | 3D Constraints | Rigid Set*
toolbar: *3D Constraints* |

: ***dmrigidset3d***
Prompts you in the command bar:

Select edges, faces, 3D entities or [WOrld/BLock]: (Select entities or subentities or choose an option.)

Entities in set: 1

Select edges, faces, 3D entities or [WOrld/BLock]: (Select more entities or subentities or choose an option, or press Enter.)

Press the TAB key to select obscured geometry.

All members of a rigid set are moved (DmMove command) and rotated (DmRotate command) together, while their relative positions within the rigid set are preserved.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmRotate

Rotates faces of a solid around an axis.

Accessing the Command

command bar: *dmrotate*
menu bar: *Model | Direct Modeling | rotate*
toolbar: *Direct Modeling |*

: ***dmrotate***
Prompts you in the command bar:

Select several entities/subentities: (Select the faces or detected boundaries you want to rotate. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)

Select edge or line as axis or define axis by <2Points>/Xaxis/Yaxis/Zaxis: (Choose an option or right click to define the axis by 2 points.)

Command Options

Option	Description
2Points	Prompts you in the command bar. Specify start point of axis <0,0,0> - Specify a point or right click to us the origin of the current coordinate system. Specify end point of axis or <use axis parallel to view direction> - Specify a point or right click to define the axis parallel to the current view direction (= perpendicular to the screen).
Xaxis	Rotates the selection around the X-axis of the current coordinate system.
Yaxis	Rotates the selection around the Y-axis of the current coordinate system.
Zaxis	Rotates the selection around the Z-axis of the current coordinate system.

Related Commands

dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmDelete - deletes faces and solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
dmMove - moves the selected faces or edges of a solid using a vector.
dmPushPull - adds or removes volume from a solid by moving a face or surface.
dmRevolve - Creates 3D solids by revolution of closed 2D entities or regions about an axis.

DmSelect

Selects edges and faces of 3D solids or surfaces based on their geometric properties.
The command allows to select a group of faces, which form a geometric feature, such as a protrusion, a depression, a blend, or a blend network. You can combine different options of this command, e.g. select all blends, which radius is less than given value.

Accessing the Command

command bar: *dmselect*
menu bar: *Model | Smart Selection*
Quad command group: *Select*

: ***dmselect***
Prompts you in the command bar:
Select [Selection/Attribute/Relation/Primitive/feaTure/Finish] <Finish>: *(Select an option.)*

If a selection set of faces or edges is present, the initial prompt is:
Use preselection as [**sUbset**/**Sample**/**sEed**] <Sample>: *(Select an option.)*

Selecting Cancel in a sub-prompt or pressing the Esc button returns to main prompt.

Command Options

Option	Description
Selection	Allows to select a set of subentities. Prompts you: Select [sUbset/Sample/sEed] <Sample>: • Subset: only faces/edges in the subset can be selected. • Sample: the preselected set of faces/edges should be considered as samples, of which the geometric characteristics (length, area, radius) will be used as parameters for the selection. • Seed: the preselected set of faces /edges be should be considered as seed elements for the selection of features such as blends, blend networks, protrusion or depressions.
Attribute	Prompts you: **Select [Radius/Area/Length/Type/Exclude short fillets] <Radius>:** • Radius: selects faces/edges by radius • Area: selects faces by area • Length: selects edges by length • Type: selects faces/edges by their geometric type If you choose any of the above options, you are promted: Select [Current sample/Value] <Current sample>: • Current Sample: Press <ENTER> to use the value of the sample face/edge • Value:type a value in the command bar In case of a non-zero value, you are prompted: Is [Less/Less or eQual(LQ)/Equal/Greater or eQual(GQ)/Greater] <Equal>: Select an option. • Exclude short fillets: excludes short fillets from the selection, which might be useful when working with imported sheet metal parts.

Relation	Prompts you: **Select [Coincident/Parallel/coAxial/Smooth] <Parallel>:** • Coincident: selects coincident faces/edges • Parallel: selects parallel faces/edges • Coaxial: selects coaxial faces/edges • Smooth: selects the smooth side of a part.
Primitive	Prompts you: Select [Face/Edge/Loop/edge Network] <Face>: • Face: selects faces • Edge: selects edges • Loop: selects edge loops Please not that the preselection will determine the result. It is necessary to select a set of connected faces. The command will then select all edges, which describe the border of the selection set; internal edges will be skipped. If the result contains multiple loops and you want only one, the preselection should contain a set of connected faces an 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 adjacent edge which have similar convexity for a given input edge
feaTure	Prompts you: **Select [Protrusion/Depression/Blend/blend Network] <blend Network>:** • **Protrusion**: selects protrusions • Depression: selects depressions • Blend: selects blends • **Blend Network**: selects blend networks
Finish	Concludes the command and selects faces/edges according to the selected options.

Related Commands

QSelect - composes a selection set using filters.
Select - selects one or more entities; can be accessed later with the Previous option.
SelectAlignedFaces - selects all faces in a model which are coplanar with a selected face.
SelectAlignedSolids - selects all solids in a model of which a face is coplanar with a selected face..
SelectConnectedSolids - selects all faces in a model which are connected to a selected face.
SelectSimilar - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE system variable

DmSelectEdges

Selects the edges of faces and solids.

Accessing the Command

command bar: *dmselectedges*
Quad cursor menu: *Direct Modeling* |

: ***dmselectedges***
Prompts you in the command bar:

Select faces or solids: (Select faces or solids)

Entities/subentities in set: 2

Select faces or solids: (Select more faces and or solids or press Enter to stop)

The edges of all selected faces and solids are selected.

Launch a command to be applied to the selection, e.g. DmFillet or DmChamfer.

Command Options

Press and hold the Shift key, then click the edges to be removed from the selection set.

Procedures

Selecting edges of faces or solids

Related Commands

DmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SmSelect - selects hard edges or form features.

DmSimplify

Simplifies the geometry and topology of 3d solid entities.

DmSimplify removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.
It is recommended to always run this command on imported 3d solid geometry.

Accessing the Command

command bar: *dmsimplify*

: ***dmsimplify***
Prompts you in the command bar:
Select entities to simplify or [Entire model/SEttings] <Entire model>: *(Select entities or press Enter to select the entire model)*

Geometry simplified

Topology simplified

Command Options

Option	Description
Settings	Allows to access the command settings Prompts you in the command bar: **Simplify [Geometry/Topology]/<Back>:** • **Geometry**:define the Geometry settings • **Topology**: define the Topology settings • **Back**: return to the default command prompt
Geometry	Prompts you in the command bar: **Select geometric simplification tolerance or [ON/OFF/Topology] <1e-7>:** Choose an option: • Type a value to set the simplification tolerance. • **ON**: switches geometrical simplification on. • **OFF**: switches geometrical simplification off. • **Topology**: define the Topology settings.
Topology	Prompts you in the command bar: Select topology simplification mode [ON/OFF/Geometry] <ON>: Choose an option: • **ON**: switches topological simplification on. • **OFF**: switches topological simplification off. • **Geometry**: define the geometrical settings.

Related Commands

Import - imports the geometry from external files into the current drawing.
dmStitch - converts a set of region and surface entities that bound a watertight area to a 3d solid.

dmAudit - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).
Erase - erases entities from the drawing. Deletes 3D solid faces and face edges.

DmStitch

Converts a set of region and surface entities that bound a watertight area to a 3d solid.
If DMREPORTPANEL= ON, BricsCAD reports the results in the Report panel instead of the command bar.

Accessing the Command

command bar: *dmstitch*
menu bar: *Model | Direct Modeling | Stitch*
toolbar: *Direct Modeling|*

: ***dmstitch***
Prompts you in the command bar:
Select surfaces to stitch or choose [validation MOde] <Auto>: *(Select regions or surfaces.)*

Entities in set: XX

Select surfaces to stitch or choose [validation MOde] <Auto>: (Select more regions or surfaces or right click.)

Created valid 3D solid.

Command Optionsd

Option	Description
Validation Mode	Allows to set the validation mode either manual or automatic. Prompts you: **Choose validation mode [Auto/Manual] <Auto>:** Select a validation mode. The command reports success or failure. In automatic validation mode, the command creates only valid 3d solid entities without free or non-manifold edges. If you want to create an invalid 3d solid or understand the reason of a failure, run the command in manual validation mode. The command then reports problems (free or non-manifold edges) and highlights them. You can either accept the invalid 3d solid by pressing Enter or refuse it by pressing Cancel. The selected validation mode is stored and re-used for future calls of the command.

Related Commands

dmExtrude - creates 3D solids by extruding closed 2D entities, regions or closed boundaries.
dmSimplify - simplifies the geometry and topology of 3d solid entities.
dmAudit - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DmTangent3d

Applies a tangent constraint between a face and a curved surface of different solids.

Accessing the Command

command bar: *dmtangent3d*
menu bar: *Parametric | 3D Constraints | Tangent*
toolbar: *3D Constraints |*

: ***dmtangent3d***
Prompts you in the command bar:
Select first subentity [WOrld/BLock]: *(Select the first cylindrical (circular or elliptical), spherical or conical surface or choose an option.)*
Select second subentity [WOrld/BLock]: *(Select the second cylindrical (circular or elliptical), spherical or conical surface or choose an option.)*

Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; Prompts you: Select block reference - Select a block reference of which the reference coordinate system will be used as an argument. Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane] - Select a reference coordinate system subentity as an argument.

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.
DmUpdate - Forces 3d constraints to update.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

DmThicken

Creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities.
The following wire entities are accepted: lines, polylines, circles, ellipses, arcs, helices, splines

Accessing the Command

command bar: *dmthicken*
menu bar: *Model | Direct Modeling | Thicken*
toolbar: *Direct Modeling |*

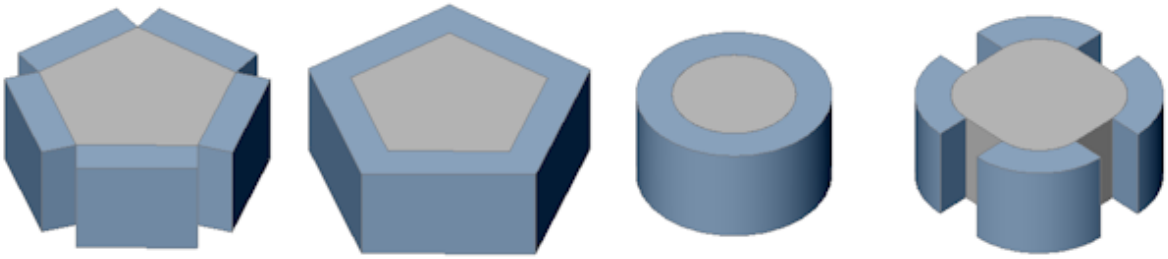
: ***dmthicken***
Prompts you in the command bar:
Select entities/subentities to thicken: *(Select a face of a solid, a surface or a wire entity.)*

Entities in set: 1

Select entities/subentities to thicken: (Select more solid faces or surfaces or press Enter.)

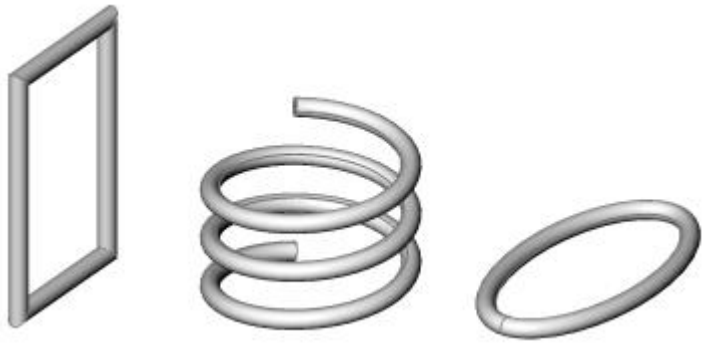
Specify thickness value: (Specify a point or type a value.)

The thickening direction is perpendicular at every point to the selected entity or entities. If you thicken a multi-face surface of several adjacent faces of one 3D solid/surface, then the corresponding faces remain adjacent on the opposite side of the new 3D solid(s). This differs the DmThicken command from DmExtrude.



From left to right: extruded faces of a prism - thickened faces of a prism - thickened face of a cylinder - thickened fillets.

When a wire entity is selected, a circle with the radius of the specified thickness value is swept along the wire:



Command Options

No command options.

Related Commands

dmExtrude - creates 3D solids or 3D surfaces by extruding closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.

DmTwist

Modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

Accessing the Command

command bar: *dmtwist*
menu bar: *Model | Direct Modeling | Deform | Twist*
toolbar: *Deform |*
ribbon: *Solid or Surface | Deform |*
Quad: *Model |*

: ***dmtwist***
Prompts you in the command bar:
Select object to twist: *(Select a solid, a surface or a region.)*

Entities in set: 1

Select object to twist: (Select more objects or right click.)

Enter start point of twisting axis: (Specify a point where the twisting deformation will start.)

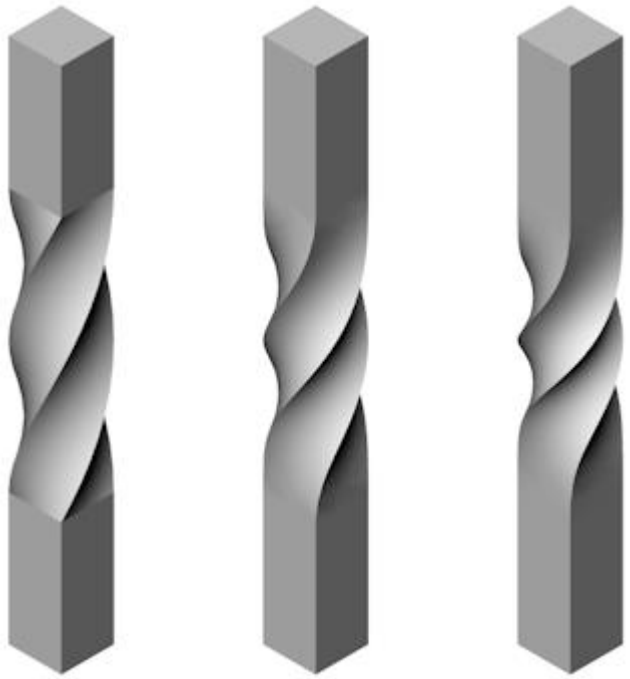
Enter end point of twisting axis: (Specify a point where the twisting deformation will end.)

Pick start point of twisting: (Specify a point.)

Specify twist angle or set [Continuity]: (Specify an angle dynamically or type a value or type C to set the Continuity.)

Command Options

Option	Description
Continuity	Allows to control whether the transition between the deformed and non-deformed part is sharp or smooth. Prompts you: Specify Continuity [G0/G1/G2] - Choose an option.

	 G0: No continuity (left), G1: Tangent Faces (middle) and G2: Curvature continuity (right)

Related Commands

dmDeformCurve - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.
dmDeformMove - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.
dmDeformPoint - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.
Loft - creates 3D solids passing through two or more cross sections.

DmUpdate

Forces 3d constraints to update.

Accessing the Command

command bar: *dmupdate*
menu bar: none
toolbar: *3D Constraints* |

: ***dmUpdate***

Command Options

No options

Related Commands

BmBrowser - displays the Mechanical Browser
DmConstraint3D - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.

DmCoincident3d - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
DmConcentric3d - applies a concentric constraint between two cylindrical, spherical or conical surfaces.
DmFix3d - Applies a fixed constraint to a solid or to an edge or a face of a solid.
DmParallel3d - applies a parallel constraint between two faces of a solid or of different solids.
DmPerpendicular3d - applies a perpendicular constraint between two faces of a solid or of different solids.
DmRigidSet3d - Allows to define a set of entities or subentities as a rigid body.
DmTangent3d - applies a tangent constraint between a face and a curved surface of different solids.

DmAngle3d - applies an angle constraint between two faces of a solid or of different solids.
DmDistance3d - applies a distance constraint between two subentities of a solid or of different solids.
DmRadius3d - applies a radius constraint to cylindrical surfaces or circular edges.

E

EAttEdit

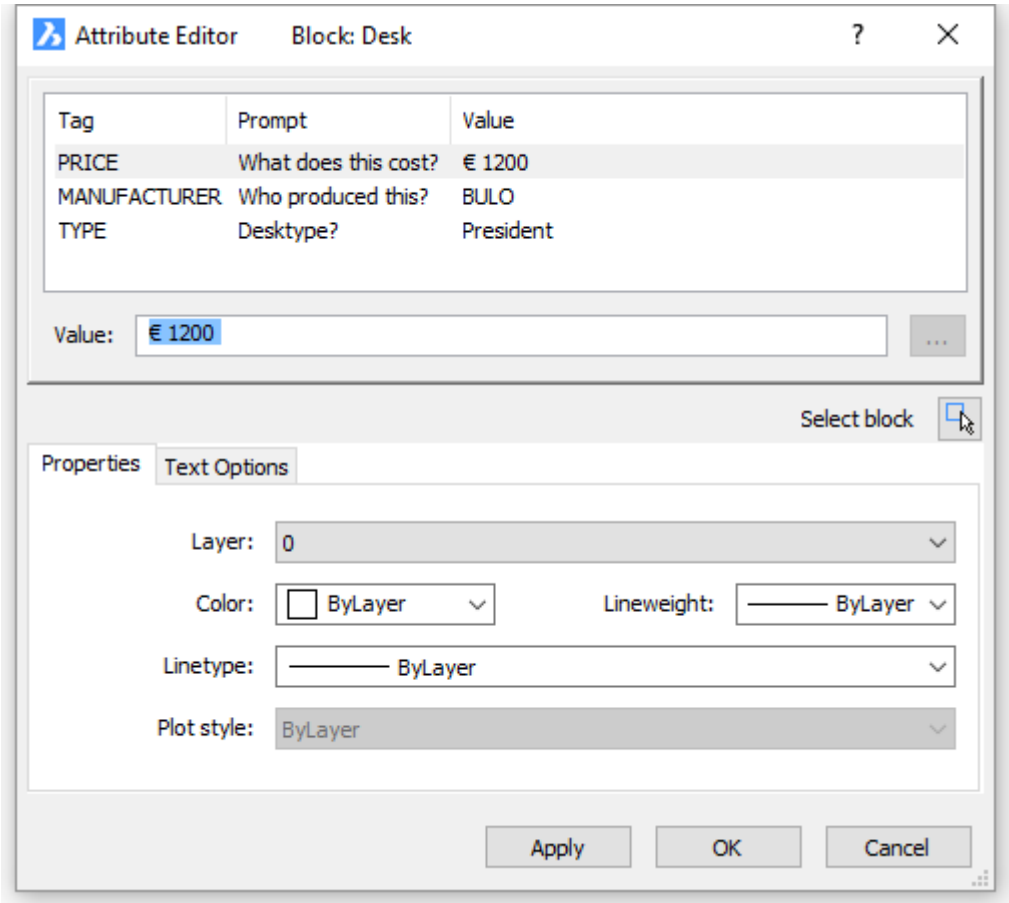
Edits the value and most properties of attributes, through a dialog box.

Accessing the Command

command bar: *eattedit*
alias: *ate*
menu bar: *Tools | Attributes | Edit Block Attributes*
toolbar: *Attributes |*

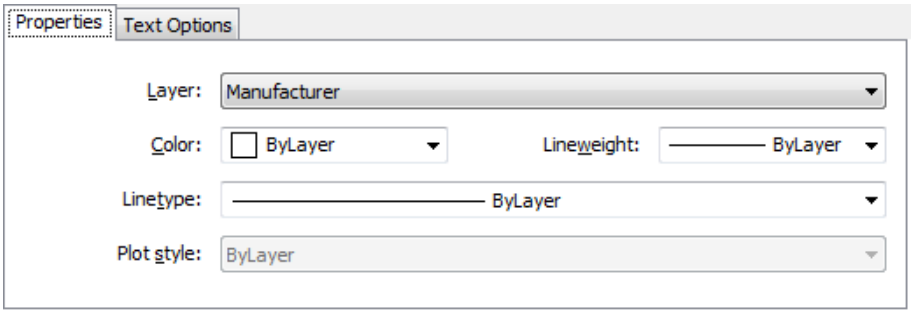
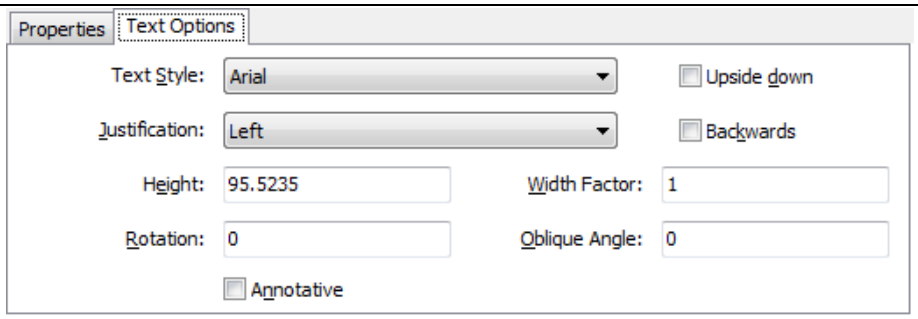
: ***eattedit***
Prompts you in the command bar:
Select a block: *(Choose a block that contains attribute data.)*

Displays a dialog box:



Change properties, and then click **OK**.

Command Options

Option	Description
Value	Edits the value of the attribute.
Select Block	Chooses another block.
Properties Tab	
Layer	Specifies the layer name; choose one from the droplist.
Color	Specifies the color; choose one from the droplist.
Lineweight	Specifies the lineweight; choose one from the droplist.
Linetype	Specifies the linetype; choose one from the droplist.
Plot Style	Specifies the plot style; choose one from the droplist. (Available only when plot styles are enabled in the drawing.)
Text Options Tab	
Text Style	Specifies the text style name; choose one from the droplist.
Justification	Specifies the justification; choose one from the droplist.
Height	Specifies the height of the text.
Rotation	Specifies the rotation angle of the text.
Upside Down	Toggles whether the text is displayed upside down.
Backwards	Toggles whether the text is displayed backwards.
Width Factor	Specifies the width of the characters; use a value of less than 1.0 to squeeze text into a small space.
Oblique Angle	Specifies the slant of the characters; can be used to mimic italic text.

Procedures

Using the Attribute Editor
Editing attributes in the Properties bar

Related Commands

DdAtte - edits the values of attributes through a dialog box.
AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttEdit - edits attribute values and properties.
-AttExt - extracts attributes from drawings to data files through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
DataExtraction - exports entity properties, block attributes and drawing information to a an external file.
Explode - reduces attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits attribute values and properties.
RefEdit - edits blocks and attributes in-place.

EdgeSurf

Creates a 3D Coons mesh surface patch between four lines, forming a closed shape (short for "edge surface").

Accessing the Command

command bar: *edgesurf*
menu bar: *Model | Meshes | Coons Surface*
toolbar: *Meshes | 3D Surfaces |*

: ***edgesurf***

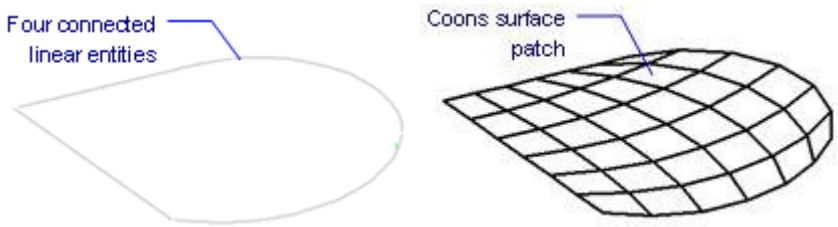
Prompts you in the command bar:

Select the first of four connected linear entities for edge surface: (Pick an entity.)

Select second edge: (Pick second entity.)

Select third edge: (Pick a third entity.)

Select last edge: (Pick a fourth entity.)



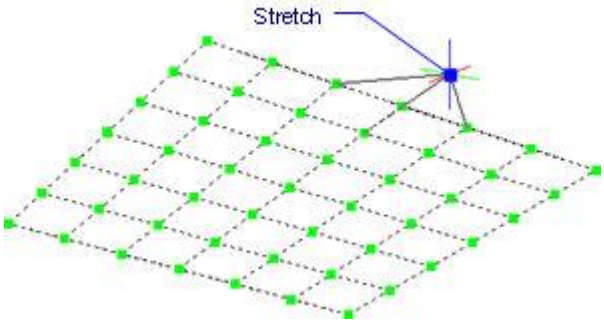
Command Options

No options.

Grips Editing

Edge meshes can be edited directly through grips:

1. Select the mesh. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

EditEData

Creates and edits extended entity data (short for "edit entity data").

Accessing the Command

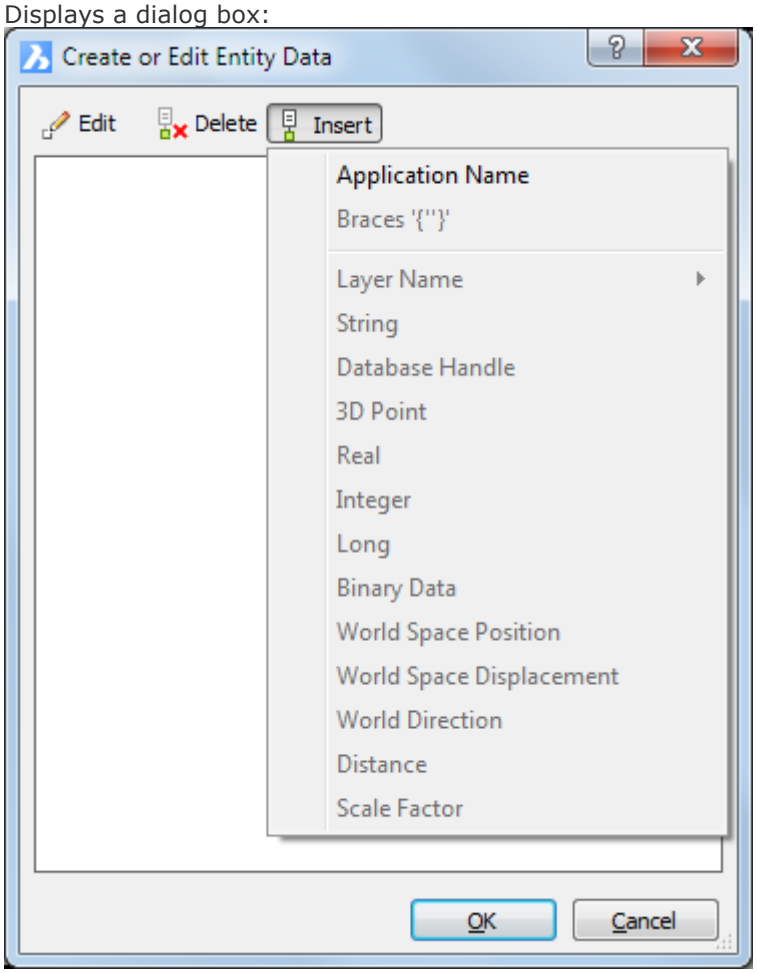
command bar: *editedata*
transparent: *'editedata*
menu bar: *Tools | Entity Data | Create or Edit Entity*
toolbar: *Entity Data |*

: ***editedata***

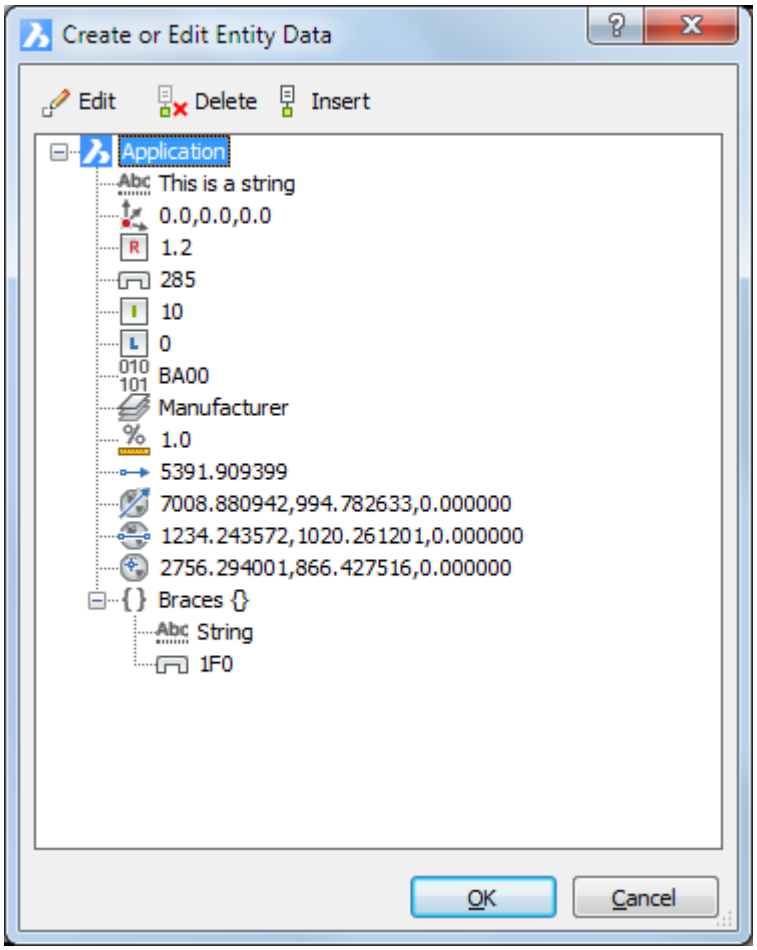
Prompts you in the command bar:

Select entity to create or edit Entity Data: (Choose any entity in the drawing.)

Displays a dialog box:



Click Insert, and the add entity data from the menu.



Dialog box after adding entity data.

Command Options

Option	Description
Select entity to create or edit Entity Data	Chooses any entity in the drawing.
Dialog Box Menu Bar	
Edit	Edits the selected parameter.
Delete	Erases the selected parameter.
Insert	Inserts extended entity data.
Application Name	Specifies the name of the application; this name identifies the "owner" of the data, and is used by other edata commands, such as DelEData.
Braces	Inserts a pair of braces {} to create subsets of data.
Layer Name	Specifies the layer; choose a layer name from the sub menu. The layer must already exist in the drawing.
String	Inserts a text string.
Database Handle	Inserts the handle (DWG database identifier) of an entity; prompts you:

	Select entity - pick any entity in the drawing.
3D Point	Inserts a 3D coordinate.
Real	Inserts a real number.
Integer	Inserts an integer (8-bit).
Long	Inserts a long (16-bit) integer.
Binary Data	Inserts binary data; useful for storing images.
World Space Position	Inserts x,y,z coordinates picked in the drawing; prompts you: Select 3D World Space Position Point - pick a point in the drawing, or enter x,y,z coordinates. "World" refers to WCS, the world coordinate space.
World Space Displacement	Inserts x,y,z coordinates picked in the drawing; prompts you: Select 3D World Space Displacement Point - pick a point in the drawing, or enter x,y,z coordinates.
World Direction	Inserts x,y,z coordinates picked in the drawing; prompts you: Select 3D World Direction Point - pick a point in the drawing, or enter x,y,z coordinates.
Distance	Inserts x,y,z distance between two points picked in the drawing; prompts you: Distance - pick a point in the drawing, or enter x,y,z coordinates. Second point - specify another point.
Scale Factor	Inserts a scale factor.

Related Commands

CopyEData - copies extended entity data to other entities.
DelEData - erases extended entity data from entities.
MoveEData - moves extended entity data to other entities.
ReAssocApp - associates applications with entities.

Elev

Changes the elevation and thickness.

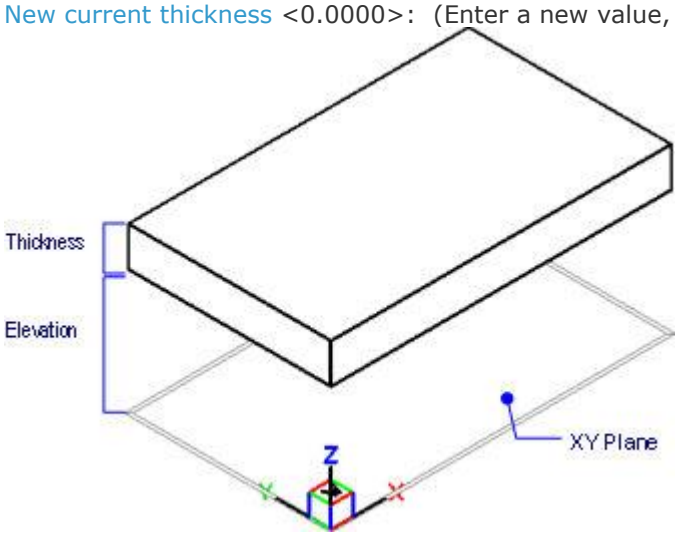
Accessing the Command

command bar: *elev*

: ***elev***
Prompts you in the command bar:

New current elevation <0.0000>: (Enter a new value, or press Enter to keep the current value.)

New current thickness <0.0000>: (Enter a new value, or press Enter to keep the current value.)



Command Options

Option	Description
New current elevation menu bar: *Settings | Elevation* toolbar: *Settings |*	Specifies the elevation; enter a negative value for a negative elevation. The elevation is reported on the status bar as the z coordinate. The elevation is the height of the xy plane in the z direction.
New current thickness menu bar: *Settings | Thickness* toolbar: *Settings |*	Specifies the thickness; the thickness begins at the elevation; enter negative values for thicknesses that are drawn downwards (negative z direction). The thickness is the extrusion distance (in the z direction) of non-3D entities, such as points, lines, polylines, arcs and circles. When thickness is not 0: • Points become lines. • Lines and open polylines become planes • Arcs become curved surfaces. • Circles become cylinders. • Closed polylines become tubes.

Related Commands

Flatten - Flattens 2D and 3D entities: projects the entities onto the XY-plane of the current view.
Change - changes the elevation and thickness of entities through the command bar.
Properties - changes the elevation (z coordinate) and elevation through a palette.
Settings - sets the default elevation and thickness through the 3D section of the Settings dialog box.

Ellipse

Draws ellipses and elliptical arcs.

Accessing the Command

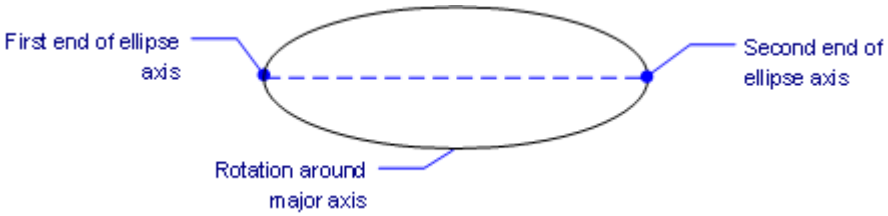
command bar: *ellipse*
alias: *el*
menu bar: *Draw | Ellipse* or *Draw | Elliptical Arc*
toolbar: *Draw |* or *Draw |*

: ***ellipse***
Prompts you in the command bar:

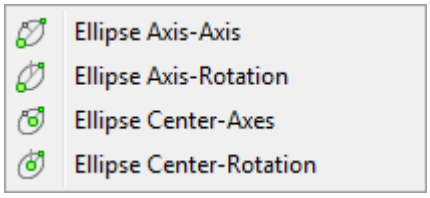
First end of ellipse axis or [Arc/Center]: (Pick a point, or enter an option.)

Second end of axis: (Pick another point.)

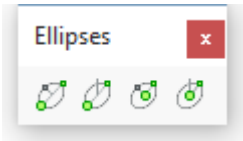
Set Other axis or [Rotation]: (Pick a third point, or enter R.)



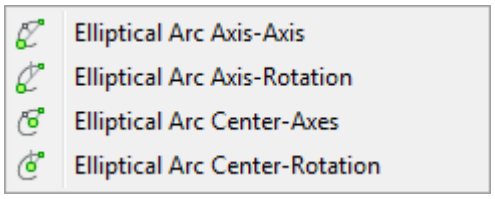
All methods to draw ellipses can be found in the Draw | Ellipse menu:



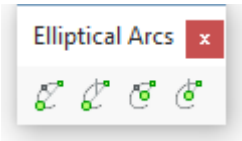
and the Ellipses toolbar:



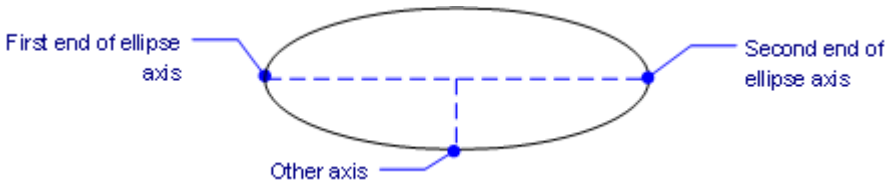
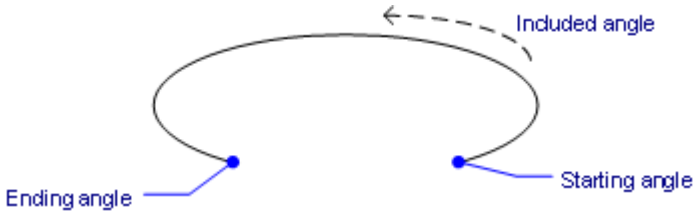
All methods to draw elliptical arcs can be found in the Draw | Elliptical Arc menu:

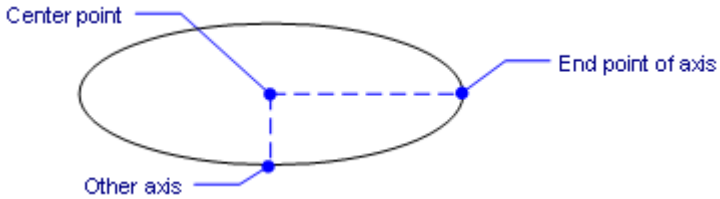
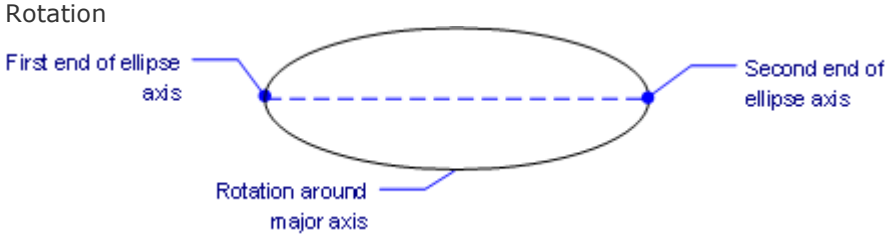


and the Elliptical Arcs toolbar:



Command Options

Option	Description
First End of Ellipse Axis 	Specifies the first endpoint of the first axis. Ellipses are defined by the length of two axes, or one axis and an angle of rotation.
Second End of Axis	Specifies the other end of the first axis.
Other Axis	Specifies the endpoint of the second axis, measured from the center of the first axis. The longer of the two axes is called the "major axis"; the shorter one is called the "minor axis."
Arc 	Draws elliptical arcs; prompts you: Center - specify the center point of the elliptical arc. First end of ellipse axis - specify the endpoint of the elliptical arc's first axis.
Parameter	Specifies parameters for the parametric formula that crafts the elliptical arc: p(b) = c + a * cos(u) = b * sin(u) where: a = major axis. b = minor axis.

	c = center of the ellipse. u = point along the arc. Cos and Sin are the trigonometric cosine and sine functions.
Start Angle of Arc	Specifies the start angle, creating the arc's starting point. The angles are measured counter-clockwise from the positive x axis.
End Angle	Specifies the end angle, creating the arc's endpoint.
Included	Specifies the included angle, measured from the starting point.
Center  	Draws the ellipse starting with a center point; prompts you: Center of ellipse - pick a point, or enter x, y coordinates.
Endpoint of Axis	Specifies the endpoint of one axis, measured from the center point.
Other Axis	Specifies the endpoint of the other axis, also measured from the center point.
Rotation  	Specifies the angle of the ellipse; prompts you: Rotation around major axis - pick a point, or enter an angle between 0 and 89.9 degrees. An angle of 0 degrees draws a circle; 89.9 draws a very thin ellipse.

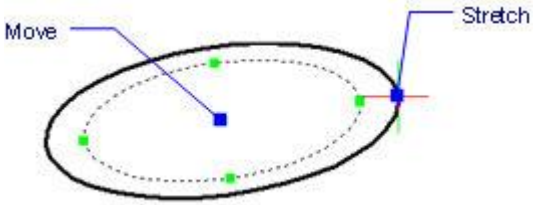
Procedures

Setting the Polyline Ellipse (PELLIPSE) variable
General procedure to draw an ellipse
General procedure to draw an elliptical arc
Editing an ellipse
Editing an elliptical arc

Grips Editing

Ellipses can be edited directly through grips:

1. Select the ellipse. Notice that it has five grips.
2. Drag the grips to perform the following actions:
    - Drag a quadrant grip to stretch the ellipse (increase or decrease the diameter).
    - Drag the center point grip to move the ellipse.



Related Commands

Arc - draws circular arcs.
Isoplane - change the working isoplane for drawing isometric entities, including isocircles.
Properties - changes the properties of ellipses through a palette.

EndCompare

Exits a drawing comparison setting, started by the DwgCompare command.

Accessing the Command

command bar: *endcompare*
toolbar: *Compare* |

: ***endcompare***
Displays no prompts you in the command bar.

Command Options

No command options.

Related Commands

DwgCompare - compares a selected drawing with the current drawing during a comparison session.
KeepMe - allows to merge the compared drawings visually.

Endpoint

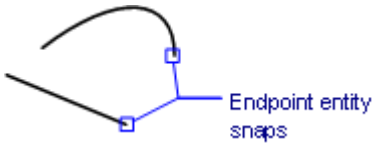
Toggles endpoint entity snap; snaps to the ends of open entities, such as line, arcs, and open polylines and vertices of 3D solids.

Accessing the Command

command bar: *endpoint*
transparent: *end*
toolbar: *Entity Snaps* |

: ***endpoint***

Displays no prompts in the command bar.



When endpoint entity snap is on, the cursor snaps to the endpoints of open entities, such as lines and arcs.

This entity snap works with 3D solid models.

Command Options

Option	Description
Endpoint	Entering the command toggles Endpoint entity snap. It snaps to the ends of open entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Erase

Erases entities from the drawing. Deletes 3D solid faces and face edges.

Accessing the Command

command bar: *erase*
shortcut: *Del*
alias: *delete, e*
menu bar: *Edit | Erase*
toolbar: *Standard |* ✕
Quad: *Model* ✕

: ***erase***
Prompts you in the command bar:

Select entities, edges or faces: (Select one or more entities.)
Entities in set: 1

Select entities, edges or faces: (Select more entities, or press Enter.)

Command Options

Option	Description
Entities	Selected entities are erased from the drawing.
Edges	Select the edges of an opening in a surface to delete the opening. Select the edge(s) between coplanar faces of a 3D solid to remove the edges. Alternatively use dmSimplify to remove redundant edges.
Faces	Select the interior faces of an opening in a 3D solid to remove the opening.

Procedures

Deleting holes
To remove edges between coplanar faces

Related Commands

Break - erases a portion of an entity.
dmSimplify - simplifies the geometry and topology of 3d solid entities.
Oops - returns erased entities.
Overkill - Deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones.
Purge - removes unused named entities from drawings, such as unused layers and linetypes.
Trim - removes portions of entities.
U - unerases erased entities.
Undo - unerases erased entities.

eTransmit

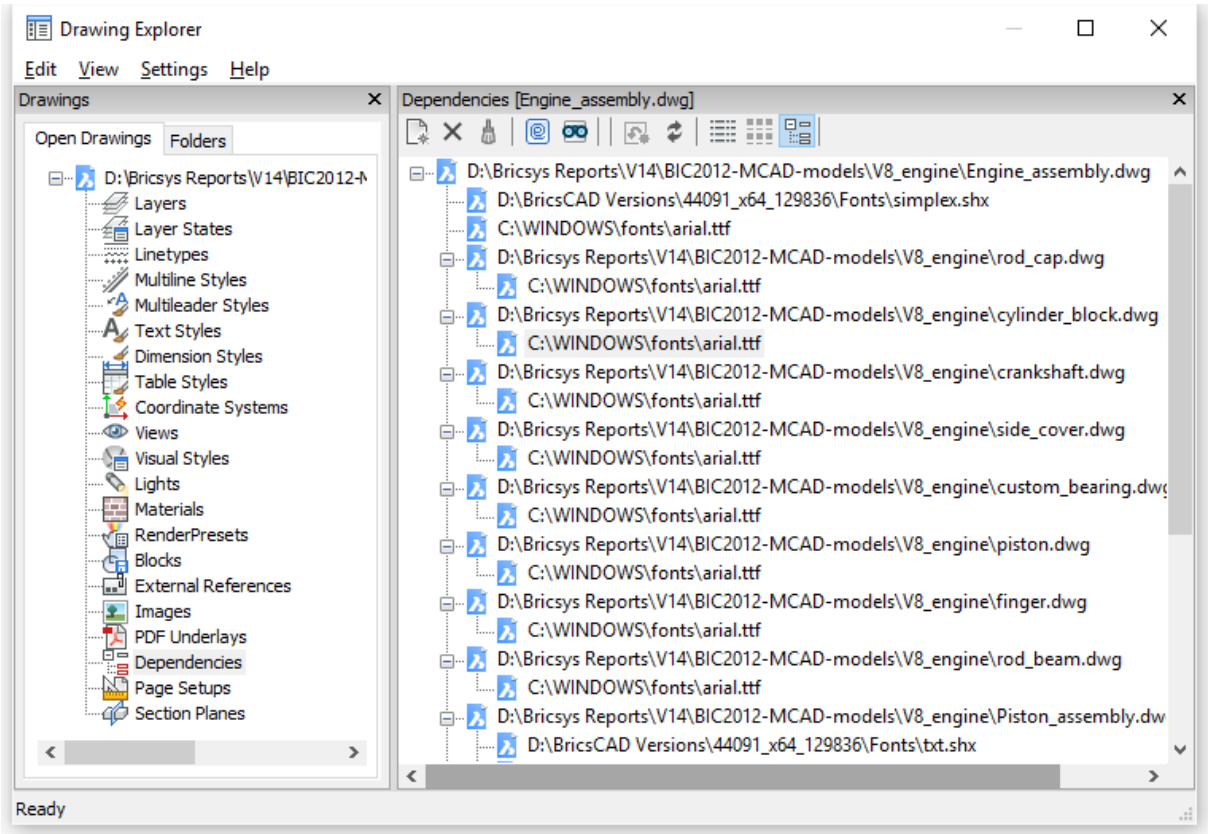
Creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

Accessing the Command

command bar: *etransmit*
alias: none
menu bar: *File| eTransmit* and *Tools | Drawing Explorer | Dependencies*

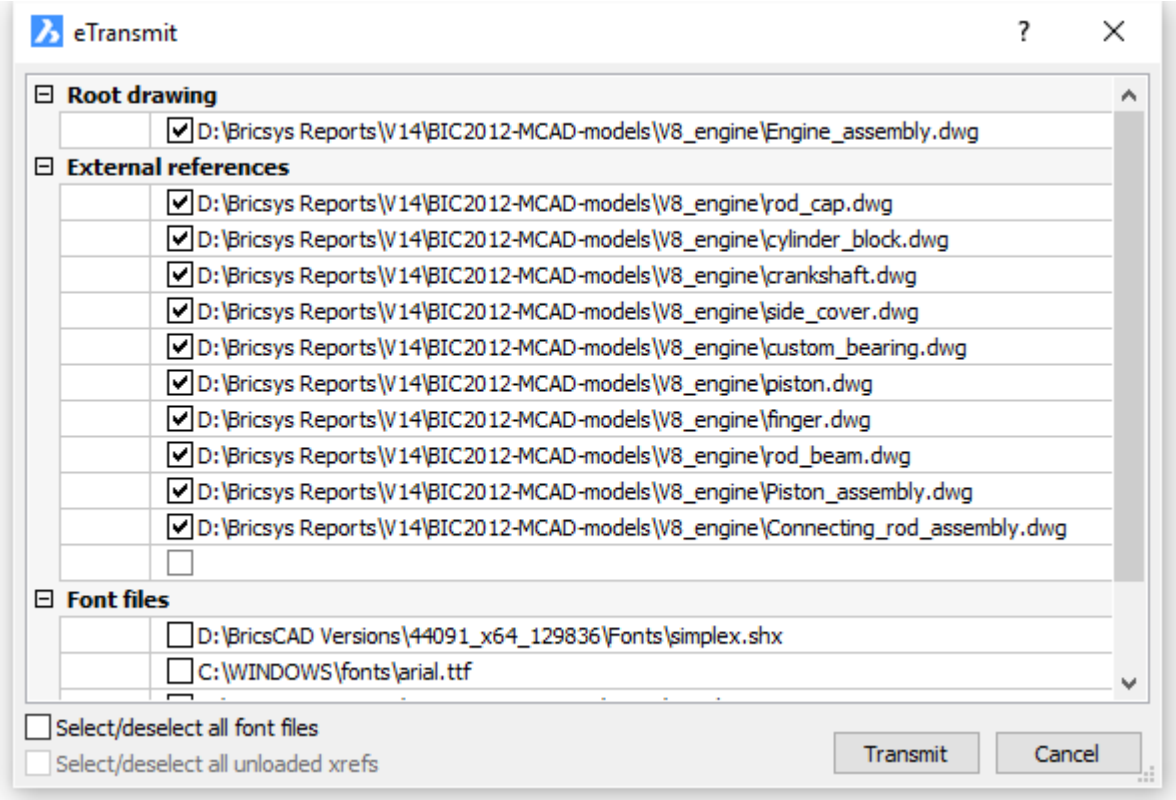
: ***etransmit***

Displays a dialog box:



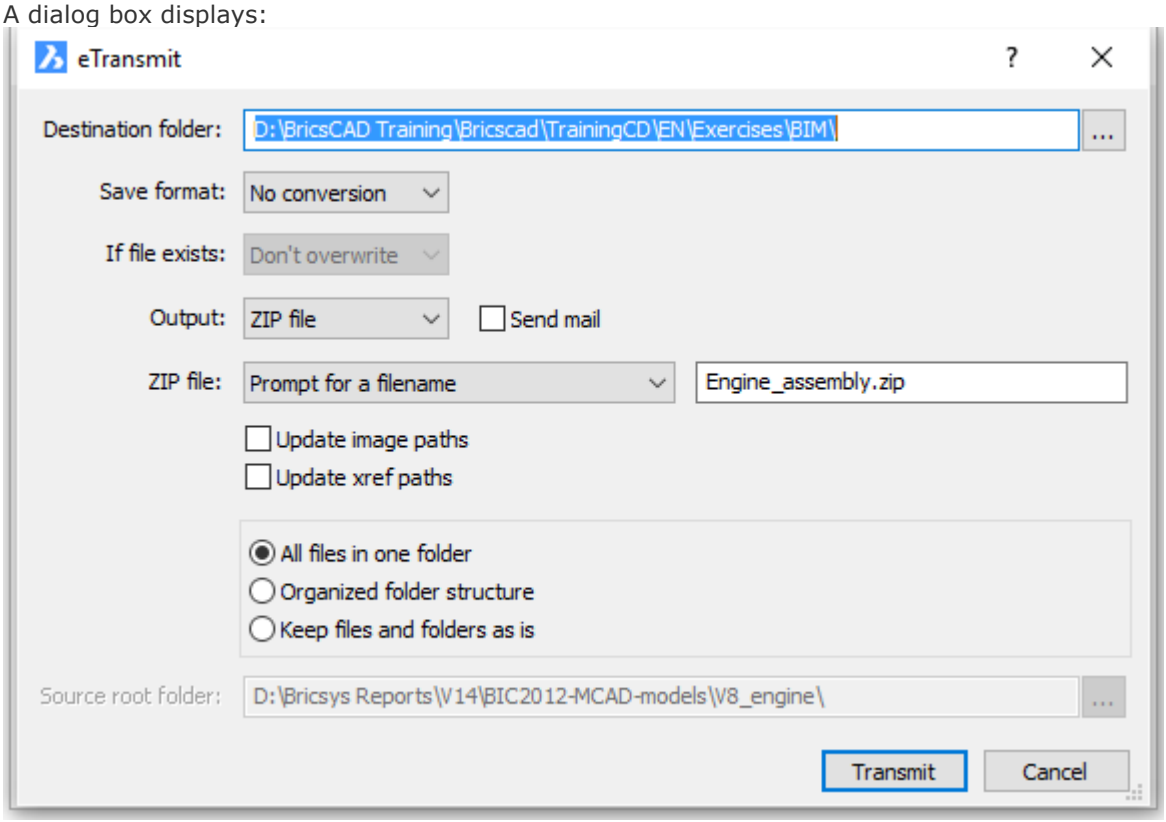
Click etransmit ( ) in the Details toolbar to start the eTransmit procedure.

A dialog box displays:



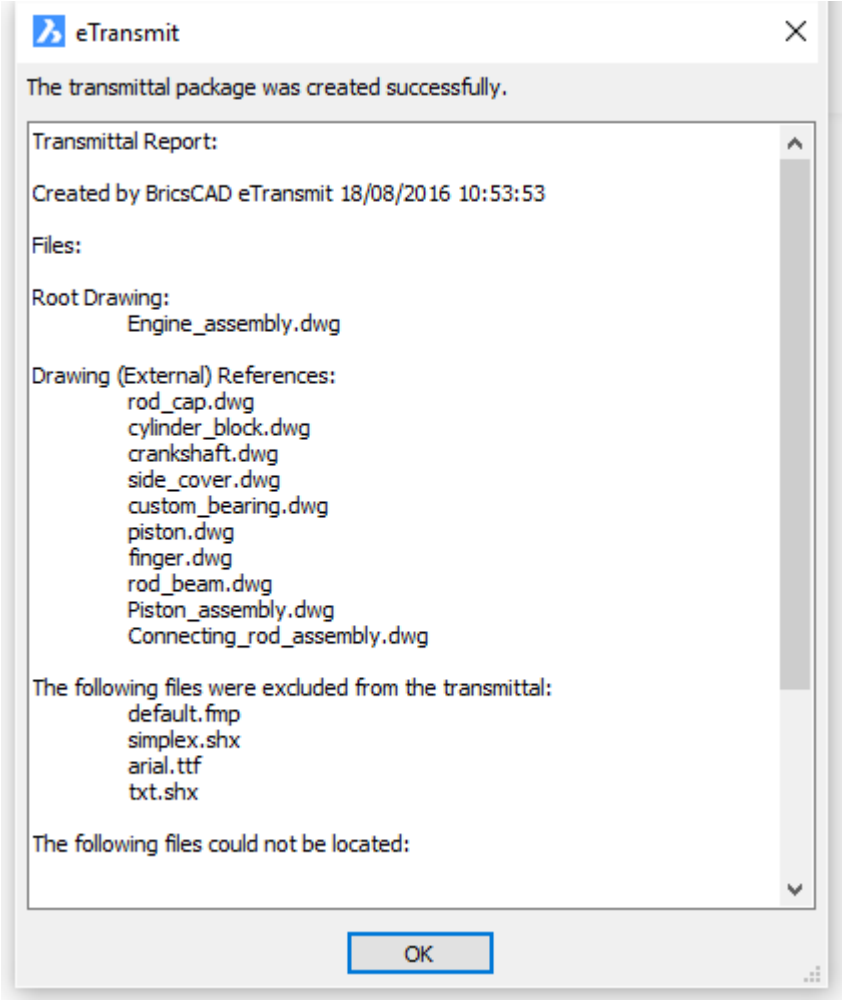
Select the files to be included in the eTransmit package, then click the Transmit button.

A dialog box displays:



Adjust the settings, then click the Transmit button.

Displays a report:



eTransmit Command Options

Option	Description
Destination folder:	Specifies the output folder for the eTransmit package. If the destination folder does not exist, you are prompted to create the folder.
Save format:	Specifies the format of the drawing files in the eTransmit package. No conversion R14 R15 (2000) R18 (2004) R21 (2007) R24 (2010) R27 (2013)
If file exists:	Specifies what to do if a file already exists in the output folder. Overwrite Don't overwrite Cancel
Output:	Specifes whether to create a ZIP file or copy the files to the output folder. Folder ZIP file

Send email:	If checked, sends an email with the zipped eTransmit package in attachment. This option is available if the eTransmit output is a ZIP file only.
ZIP file:	Specifies what to do if a ZIP file of the same name already exists in the output folder. Prompt for a filename Overwrite if necessary Increment name if necessary
Update image paths	If checked, the saved paths for image files are updated, according to the new location.
Update xrefs paths	If checked, the saved paths for xrefs are updated, according to the new location.
All files in one folder	All files in the eTransmit package are copied to the destination folder.
Organized folder structure	A hierarchical folder tree is created in the destination folder
Keep files and folders as is	Keeps the folder structure of all files in the eTransmit package. This procedure allows to easily install the package on another computer using the folder structure of the source system.
Source root folder:	Specifies the source root folder for relative paths of drawing-dependent files, such as xrefs or image files.

Procedures

To use Etransmit

Related Commands

CloudUpload - uploads the current drawing to Bricsys 24/7.
Explorer - Opens the Drawing Explorer dialog box.

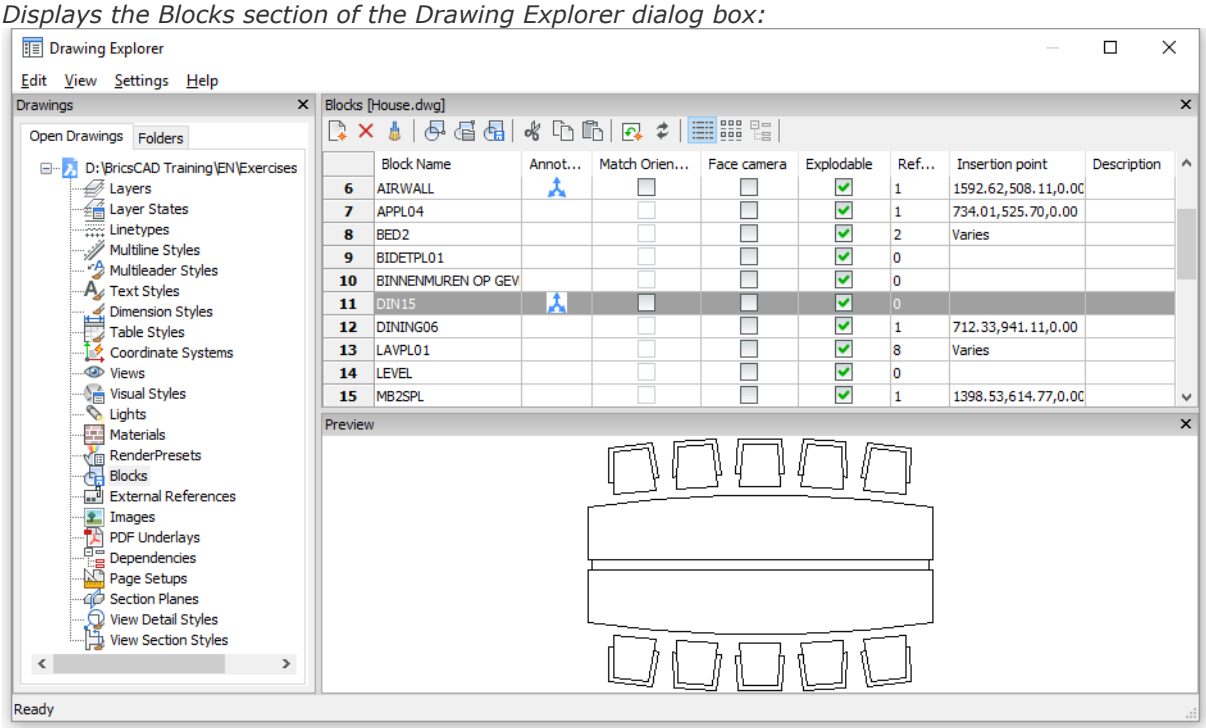
ExpBlocks

Opens the Blocks section of the Drawing Explorer dialog box (short for "explorer blocks").

Accessing the Command

command bar: *expblocks*
alias: *xb, explorer*
menu bar: *Tools | Drawing Explorer | Blocks*

: ***expblocks***
Displays the Blocks section of the Drawing Explorer dialog box:



Create or edit blocks, and then click **X**.

*Press **F1** to access help.*

Command Options

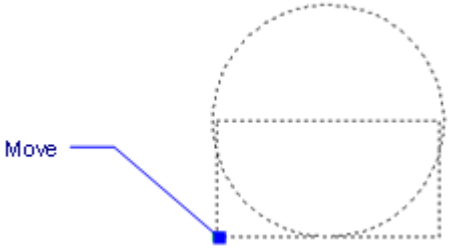
Option	Description
Block Name	Specifies the name of the block; click to rename.
Annotative	If checked, sets the Annotative property.
Match Orientation	If checked, sets the Match Orientation property of Annotative blocks.
Face Camera	
Explodable	If checked, blocks of this definition can be exploded into its component parts by the Explode command.
References	Reports the number of insertions of the block.
Insertion Point	Reports the insertion point of the block; reports "Varies" when the block has more than one insertion.
Description	Description of the block.

Toolbar Buttons	
New	Creates new blocks; dismisses the Drawing Explorer dialog box, and the starts the Block command. See Block command.
Delete	Deletes the block definition. (Available only for block definitions not inserted in the drawing.)
Purge	Purges selected blocks from the drawing. (Available only for block definitions not inserted in the drawing.)
Insert	Inserts the selected block; dismisses the Drawing Explorer dialog box, and the starts the -Insert command. See -Insert command.
Insert External Blocks	Inserts DWG files as blocks; starts the -Insert command, and then displays the Insert Block dialog box. See -Insert command.
Save Block	Exports the selected block as a DWG file; starts the WBlock command, and then displays the Save Block dialog box. See WBlock command.

Grips Editing

Blocks can be edited directly through grips:

1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.



Procedures

Open the Blocks Explorer
Open the Detail View of the Blocks Explorer
Open the Icon View of the Blocks Explorer
To rename a block
To insert a block from an unopened drawing

Related Commands

Insert - inserts blocks through a dialog box.
-Insert - inserts blocks through the command bar.
Block and -Block - creates blocks at the command line.

Explorer - Opens the Drawing Explorer dialog box.

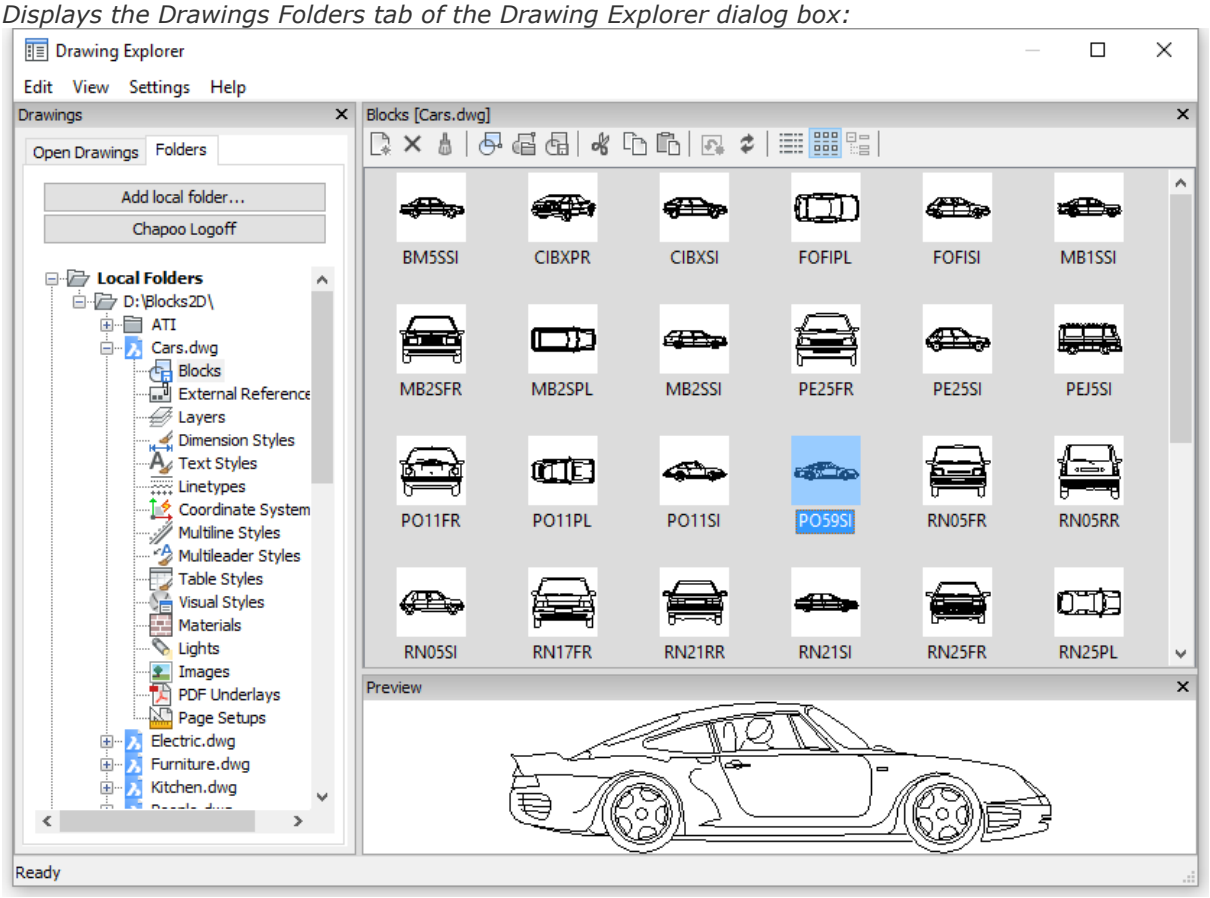
ExpFolders

Opens the Drawing Explorer on the Folders tab.

Accessing the Command

command bar: *expfolders*

: ***expfolders***
Displays the Drawings Folders tab of the Drawing Explorer dialog box:



Command Options

No options

Procedures

Adding a drawing folder
To insert a block from an unopened drawing

Related Commands

Explorer - Opens the Drawing Explorer dialog box.
ChapooLogon - Logs on to Chapoo.
ChapooOpen - Downloads and opens a drawing from Chapoo.

Explode

Reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs.

Accessing the Command

command bar: *explode*
menu bar: *Modify | Explode*
toolbar: *Modify |*

: ***explode***
Prompts you in the command bar:

Select entities to explode: (Choose one or more entities.)
Entities in set: 1

Select entities to explode: (Choose more entities, or press Enter to exit command.)

Exploded polylines become lines and arcs; width is lost.

Exploded dimensions becomes lines, text, and solids (arrowheads).

Exploded blocks become individual entities and nested blocks. (You may need to use Explode a second time.)

ByBlock colors and linetypes may change.

Attributes revert to attribute definitions.

Nonuniformly-scaled blocks are not exploded, unless the *EXPLMODE* system variable is turned on.

3D surface entities are exploded to 3D faces.

Command Options

Option	Description
Select entities to explode	Choose one or more entities. Entities on frozen and locked layers cannot be exploded.

Related Commands

Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
Xplode - explodes complex entities, with control over the results.
TxtExp - explodes text into polylines.
U - undoes the effect of the explosion.

Explorer

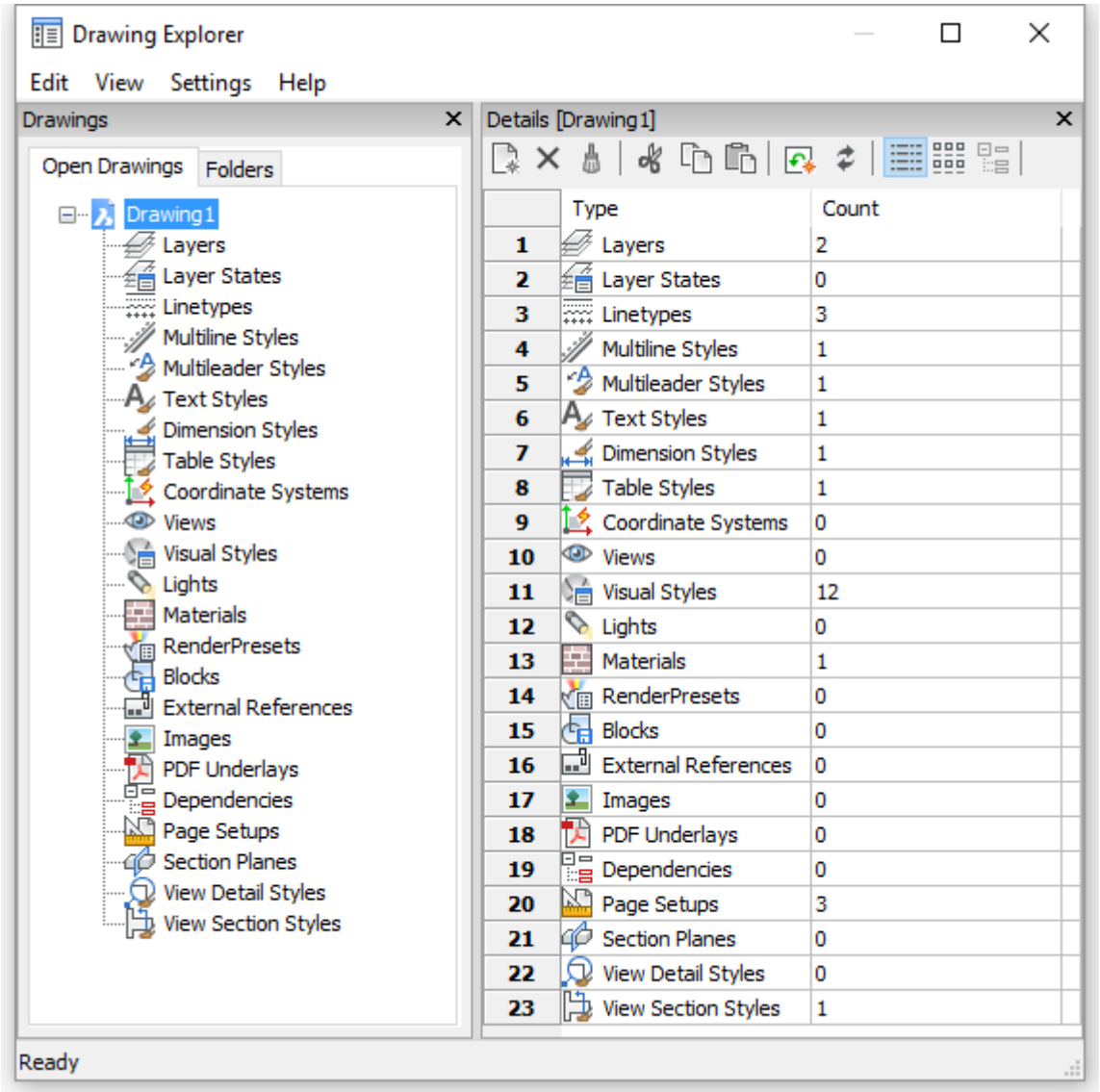
Opens the Drawing Explorer dialog box; controls Layers, Layer States, Linetypes, Multiline Styles, Text Styles, Dimension Styles, Table Styles, Coordinate Systems, Views, Visual Styles, Lights, Materials, Render Presets, Blocks, External References, Images, Pdf Underlays, Dependencies, Page Setups and Section Planes.

Accessing the Command

command bar: *explorer*
menu bar: *Tools | Drawing Explorer | (select a category)*
toolbar: *Standard |*

: ***explorer***

Displays a dialog box:



Press F1 to access help.

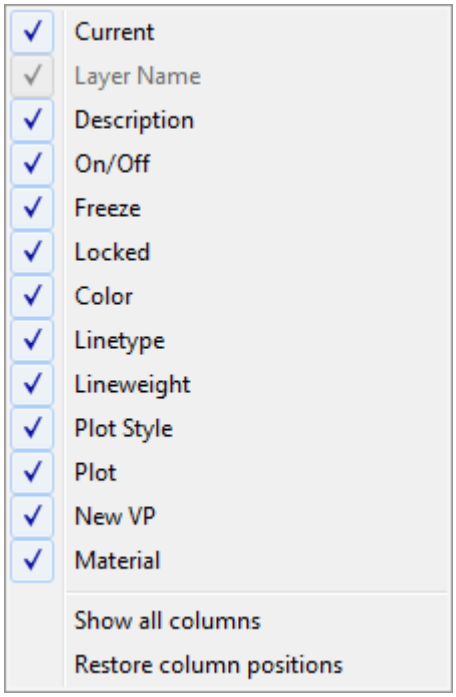
Click X to close.

Command Options

Columns of grid views, such as the Layer Explorer detail view, can be switched On/Off and repositioned by drag-and-drop.

To control the display of columns:

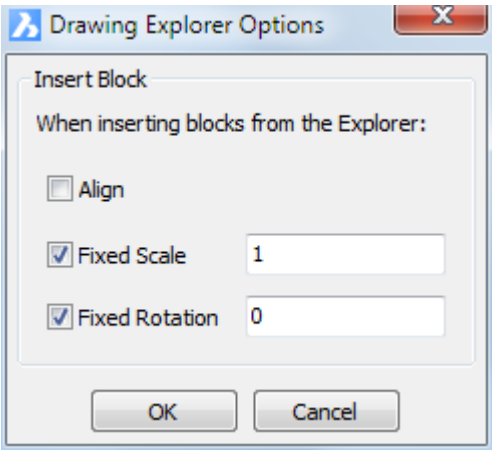
- Select a column, then right click.
  A context menu displays:



- Click a check box to display/hide a column.
- Click the *Show all columns* option to display currently hidden columns.
- Click the *Restore column positions* option to move all columns to their default position.

Edit Menu / Icon	Shortcut	Description
New	Ctrl+N	Creates a new item.
Delete	Ctrl+D	Erases the selected item.
Merge to ...		Merges the content of the selected layer to another layer.
Purge	Ctrl+P	Purges the unused items.
Cut	Ctrl+X	Copies the item to the Clipboard, and then purges it from the drawing.
Copy	Ctrl+C	Copies the item to the Clipboard.
Paste	Ctrl+V	Copies the item from the Clipboard.
Rename	F2	Renames the selected item.
Select All	Ctrl+A	Selects all items.
Current		Indicates the currently loaded item.

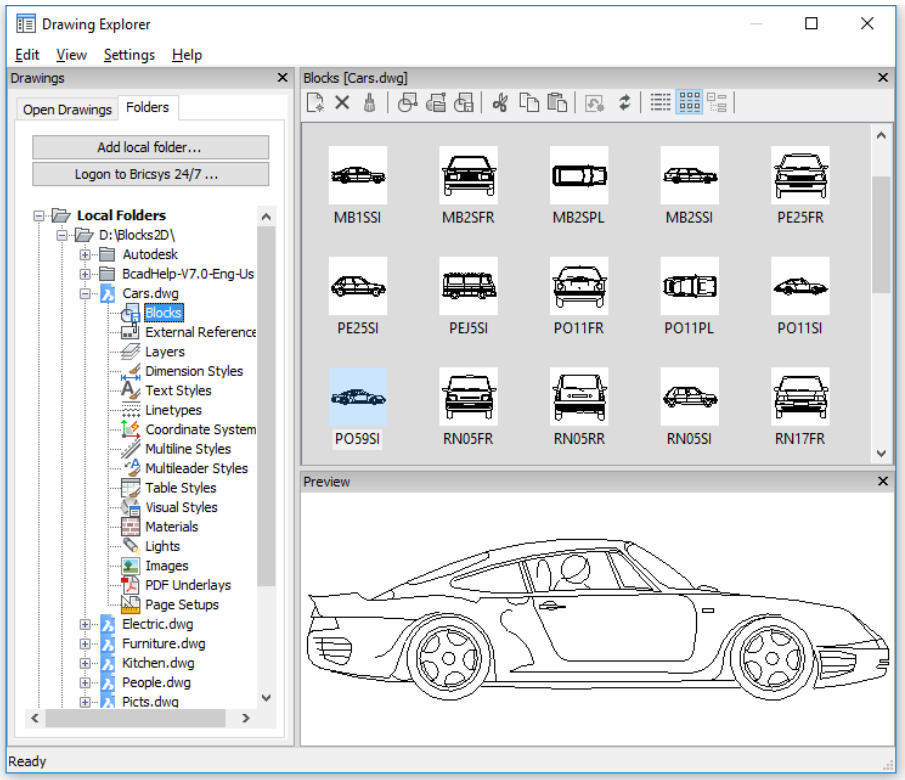
View Menu / Icon	Description
Open Drawings	If checked, displays the *Drawings* pane.
Details	If checked, displays the *Details* pane.
Preview	If checked, displays the *Preview* pane.
Editor	If checked, displays the *Edit* pane.
Status Bar	Toggles the display of the Status Bar of the *Drawing Explorer* dialog box.
Details	Displays details of items.
Icons	Displays items as icons.
Tree	Displays items in tree form (applies to layer states, dimension styles, external references, images, pdf underlays and dependencies only).
Regen	Updates the drawing display to reflect changes made in this dialog box.
Toggle Regen	Toggles automatic updating of the display.
Settings Menu	
Restore Explorer Default Layout	Restores the layout of the Drawing Explorer to the default layout for the selected category. Changes to the Drawing Explorer dialog layout are remembered for each category.
Drawings	
Open Drawings	Displays a list of drawings that are currently open. Click a drawing to see a detailed list of its content in the Details pane.
Folders	Displays a list of your favorite drawing folders. • Click the *Add Local Folder* button to add a folder, including all subfolders. • Double click a drawing to open it. • Drag & Drop definitions between unopened drawings which allows copying Layers, TextStyles, LineTypes and Coordinate systems from one drawing to another. • Block Manager: Click the *Expand* icon (+) in front of drawing, then select *Blocks* (Blocks) to see all blocks in the drawing. When in *Icon View* ( ), double click a block to place the block in the current drawing. Choose *Options* in the *Settings* menu to control the insertion procedure.



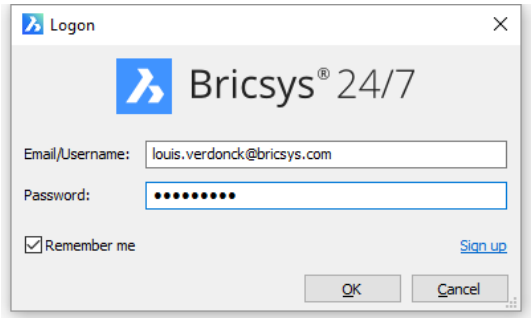
- *Align*: If checked, launches the InsertAligned command to insert the block.
  When the *Align* option is checked, the *Fixed Scale* and *Fixed Rotation* options are not available.
- *Fixed Scale*: If checked, allows to insert the block at a fixed scale.
- *Fixed Rotation*: If checked, allows to insert the block at a fixed rotation.

TUTORIALS

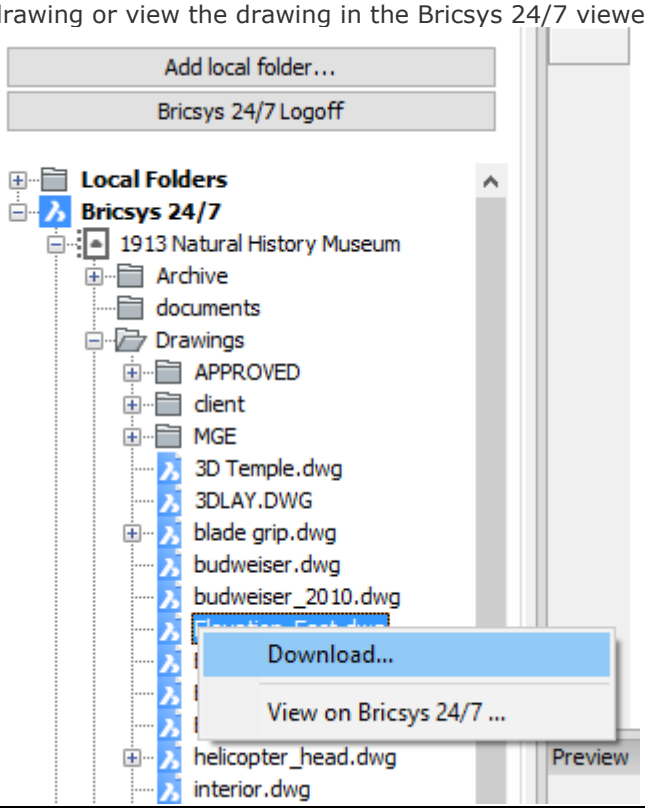
Manage blocks with the Drawing Explorer



- Click the *Logon to Bricsys 24/7* button to get access to drawings in your Bricsys 24/7 projects.



Select a drawing in a project, then right click to either download the drawing or view the drawing in the Bricsys 24/7 viewer.



Details	
Layers	Indicates the number of layers in the selected drawing. Click to explore the layers (see the Layer command).
Layer States	Indicates the number of layer states in the selected drawing. Click to explore the layer states (see the LayerState command).
Linetypes	Indicates the number of linetypes in the selected drawing. Click to explore the linetypes (see the Linetype command)
Multiline Styles	Indicates the number of multiline styles in the selected drawing. Click to explorer the multiline styles (see the MStyle command).
Multileader Styles	Indicates the number of multileader styles in the selected drawing. Click to explore the multileader styles (see the MLeaderStyle command).
Text Styles	Indicates the number of text styles in the selected drawing. Click to explore the text styles (see the Style command).
Dimension Styles	Indicates the number of dimension styles in the selected drawing. Click to explore the dimension styles (see the DimStyle command).
Table Styles	Indicates the number of table styles in the selected drawing.

	Click to explore the table styles (see the TableStyle command).

Coordinate Systems	Indicates the number of user coordinate systems (UCS) in the selected drawing. Click to explore the coordinate systems (see the ExpUcs command).
Views	Indicates the number of saved views in the selected drawing. Click to explore the saved views (see the View command).
Visual Styles	Indicates the number of visual styles in the selected drawing. Click to explore the visual styles (see the VisualStyles command).
Lights	Indicates the number of lights in the selected drawing. Click to explore the lights (see the LightList command).
Materials	Indicates the number of materials in the selected drawing. Click to explore the materials (see the Materials command).
RenderPresets	Indicates the number of user defined render presets in the drawing. Click to explore the render presets (see the RenderPresets command)
Blocks	Indicates the number of blocks in the selected drawing. Click to explore the blocks (see the ExpBlocks command).
External References	Indicates the number of external references (Xrefs) in the selected drawing. Click to explore the Xrefs (see the Xref command).
Images	Indicates the number of images in the selected drawing. Click to explore the images (see the Image command).
PDF Underlays	Indicates the number of pdf underlays in the selected drawing. Click to explore the pdf underlays (see the Pdf command).
Dependencies	Indicates the number of dependencies in the selected drawing. Click to explore the dependencies (see the eTransmit command).
Page Setups	Indicates the number of page setups in the selected drawing. Click to explore the page setups (see the PageSetup command).
Section Planes	Indicates the number of section planes in the selected drawing. Click to explore the section planes (see the SectionPlaneSettings command).
View Detail Styles	Indicates the number of view detail styles in the selected drawing. Click to explore the view detail styles (see the ViewDetailStyle command).
View Section Styles	Indicates the number of view section styles in the selected drawing. Click to explore the view section styles (see the ViewSectionStyle command).

Procedures

To open the Drawing Explorer
To control the Drawing Explorer layout
Opening a drawing
Adding a drawing folder
To insert a block from an unopened drawing
To copy definitions between drawings
To merge the content of two layers

Related Commands

DimStyle - creates and modifies dimension styles, through the Drawing Explorer.
eTransmit - creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.
ExpBlocks - opens the Blocks section of the Drawing Explorer dialog box.
ExpFolders - opens the Drawing Explorer on the Folders tab.
ExpUcs - creates, modifies, and deletes named UCSs through the Drawing Explorer.
Image - inserts images in drawings through the Drawing Explorer.
LineType - Loads, sets, and creates linetypes through the Drawing Explorer.
Layer - creates layers, changes their properties, and removes layers through the Drawing Explorer.
LayerState - creates and controls layer states in the Drawing Explorer.
LightList - displays the Drawing Explorer - Lights dialog window.
Materials - creates materials and edits their properties through the Drawing Explorer.
MLeaderStyle - Creates and manages multileader styles through the Drawing Explorer.
MStyle - creates and edits multiline styles.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.
Pdf - attaches PDF files to the current drawing through the Drawing Explorer.
RenderPresets - allows to create and edit render presets, and to set the current render preset through a dialog box.
SectionPlaneSettings - Defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.
SheetSet - Manages sheet sets.
Style - creates and modifies text style with the Drawing Explorer.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.
View - creates and sets named views through the Drawing Explorer.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.
Xref - attaches DWG files to the current drawing through the Drawing Explorer.

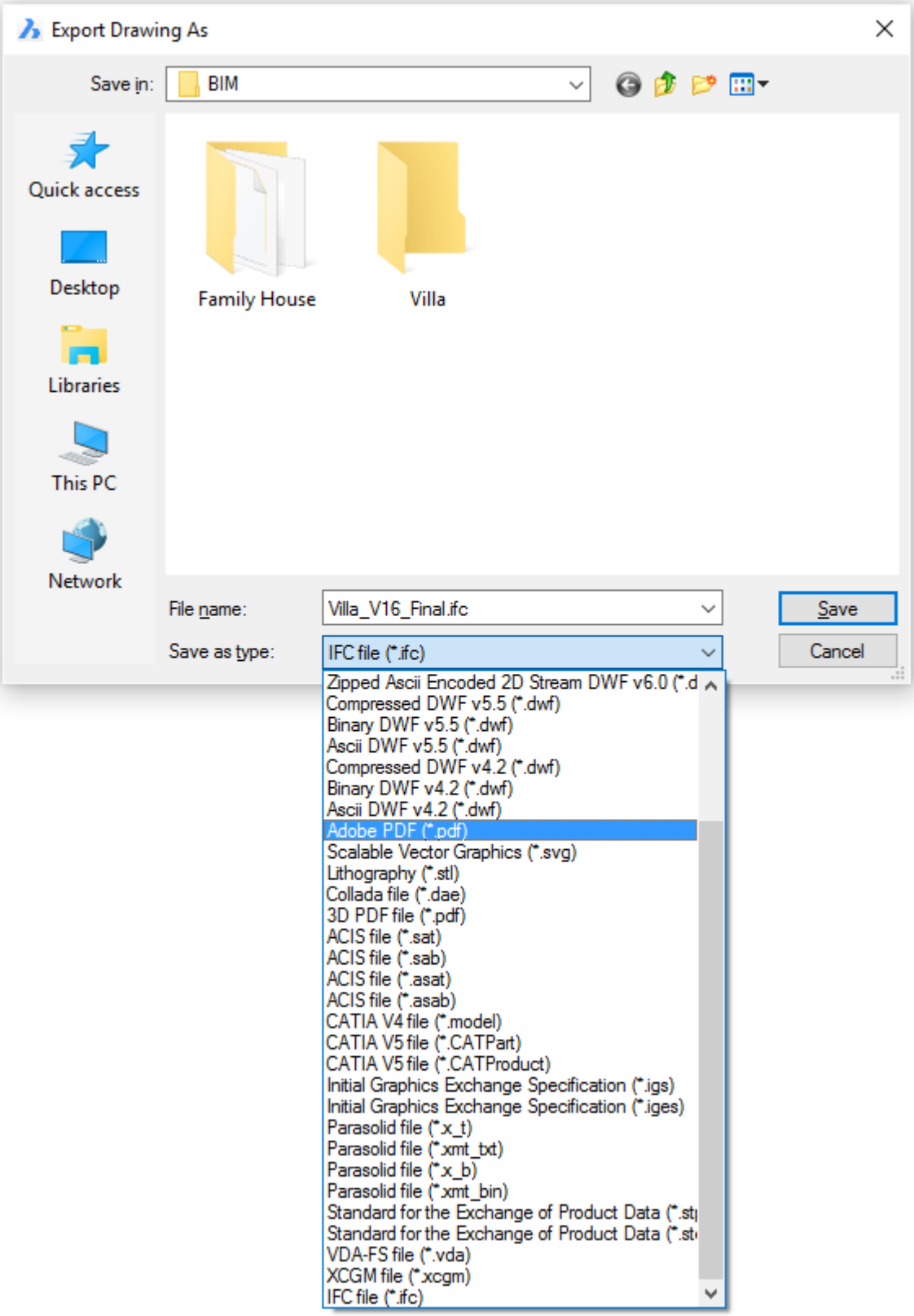
Export

Exports drawings in a variety of vector and raster formats.
Exporting to *wmf* and *emf* is not supported on Linux.

Accessing the Command

command bar: *export*
alias: *exp, dwfout*
menu bar: *File | Export*

: ***export***
Displays a dialog box:



Choose an export format from the Save As Type droplist, and then click Save.

Command Options

CAD Formats	Extension	Description
AutoCAD Drawing	DWG	CAD drawing format created by Autodesk; to export drawings in earlier version of DWG, use the SaveAs command.
AutoCAD ASCII DXF	DXF	Drawing interchange format created by Autodesk; to create binary DXF files, use the DxfOut command.
Drawing Template	DWT	Templates on which new drawings are based.
DWF	DWF	CAD viewing format created by Autodesk; short for "design Web format."
Binary DWF	DWF	Compressed version of DWF.
IFC file	IFC	Saves a BIM model as an IFC file for use with an IFC-certified applications. The Industry Foundation Classes (IFC) data model is intended to describe building and construction industry data. BIM properties are included in IFC export.
Vector Formats	**Extension**	**Description**
Windows Meta File*	WMF	Vector format created by Microsoft; used by the Clipboard.
Enhanced Windows Meta File*	EMF	Updated version of WMF.
Adobe PDF	PDF	Document publishing format created by Adobe; short for "portable document format."
Raster Format	Extension	**Description**
Bitmap*	BMP	Raster format created by Microsoft; used by the Clipboard.
XML Format	Extension	**Description**
Scalable Vector Graphics	SVG	Scalable Vector Graphics (SVG) is a family of specifications of an XML-based file format for describing two-dimensional vector graphics, both static and dynamic (i.e. interactive or animated). The SVG specification is an open standard that has been under development by the World Wide Web Consortium (W3C) since 1999.
Collada	DAE	COLLADA (from collaborative design activity) is an interchange file format for interactive 3D applications. It is managed by the nonprofit technology consortium, the Khronos Group, and has been adopted by ISO as a publicly available specification, ISO/PAS 17506.

CAM Format	Extension	Description
Stereolithografy	STL	STL (STereoLithography) is a file format native to the stereolithography CAD software created by 3D Systems. STL is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common, since they are more compact. See also the StlOut command.

(*) The *PictureExportScale* user preference sets the view size when exporting to picture formats (WMF, EMF, BMP).

The following 3D formats are available through a separate module, BricsCAD Communicator. Download the *Communicator* installer from the Bricsys website.

Format	Extension	Description
Standard for Exchange of Product data	STE, STP, STEP	STEP (Standard for the Exchange of Product Data) is an ISO standard exchange format used for representing three-dimensional data in a format that can be recognized by multiple programs.
Initial Graphics Exchange Specification	IGS, IGES	A file format which defines a vendor neutral data format that allows the digital exchange of information among Computer-aided design (CAD) systems.
CATIA V4	model	CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes.
CATIA V5	CATPart, CATProduct	
Parasolid	XMT_TXT, XMT_BIN X_T, X_B	Parasolid is a geometric modeling kernel originally developed by ShapeData, now owned by Siemens PLM Software (formerly UGS Corp.), that can be licensed by other companies for use in their 3D computer graphics software products.
VDA-FS	VDA	VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name is an abbreviation of "Verband der Automobilindustrie - Flächenschnittstelle", which translates to the "automotive industry association - surface data interface". Standard was specified by the German organization VDA.
3D PDF	PDF	Portable Document Format (PDF) is a file format used to represent documents in a manner independent of application software, hardware, and operating system.

Procedures

Export to PDF settings
Exporting a drawing to PDF

Related Commands

AcisOut - exports 3D solids and surface entities in SAT format.
BimExport - saves a BIM model as an IFC file for use with an IFC-certified applications.
BmpOut - exports drawings in BMP format.
CommunicatorInfo - checks whether the Communicator product is correctly installed and prints a diagnostic report in the Prompt History window.
DwfOut - exports drawings in DWF format.
ExportLayout - exports all visible objects from the current layout to the model space of a new drawing.
ExportPdf - exports layouts in a drawing to a pdf docucment.
Import - imports CAD files in a variety of formats.
MSlide - exports drawings in WMF, EMF, and SLD formats.
StlOut - exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing).
VmlOut - exports drawings in VML format.
VSlide - displays files in WMF, EMF, and SLD formats.
WmfOut - exports drawings in WMF, EMF, and SLD formats.

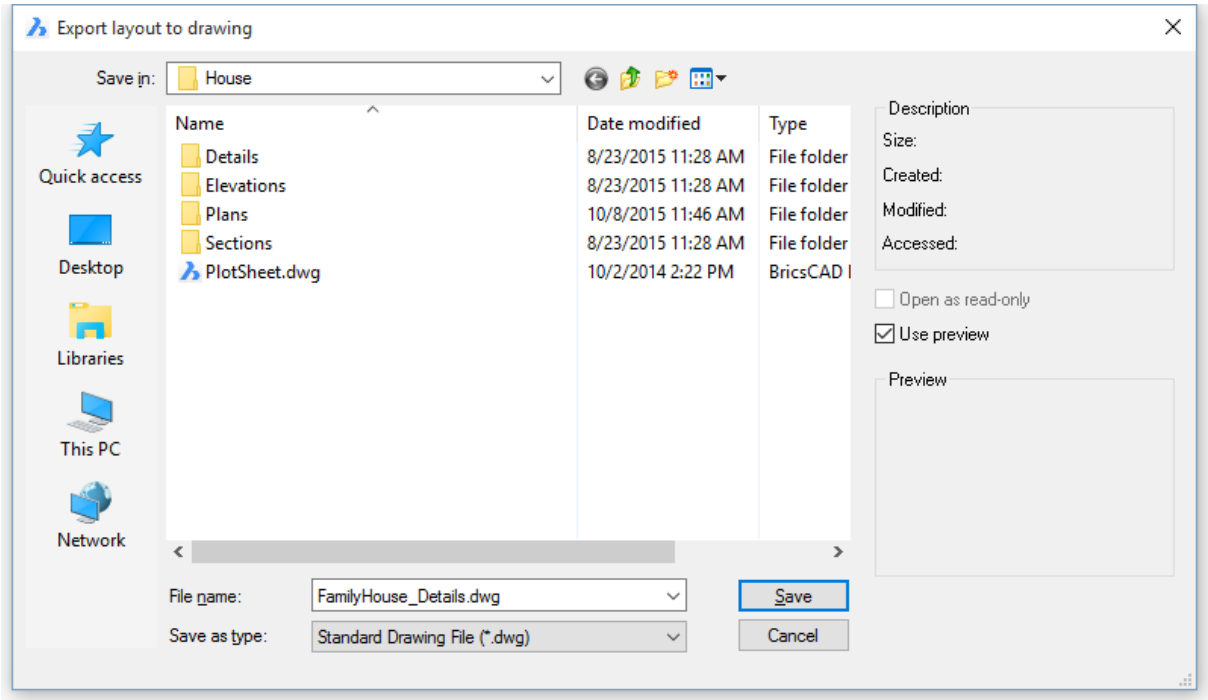
ExportLayout

Exports all visible objects from the current layout to the model space of a new drawing.
The command can only be used in a layout.

Accessing the Command

command bar: *exportlayout*
menu: *File | Export Layout to Model...*

: ***exportlayout***
Displays a File dialog box:



Type a name in the *File name* field, then press the *Save* button.

Viewports are converted to polylines.
Entities are trimmed by the viewport boundary.

Related Commands

Export - exports the drawing in a variety of raster and vector formats.
WBlock - writes blocks and other drawing parts to files on disk through a dialog box.
-WBlock - writes blocks and other drawing parts to files on disk through the command bar.

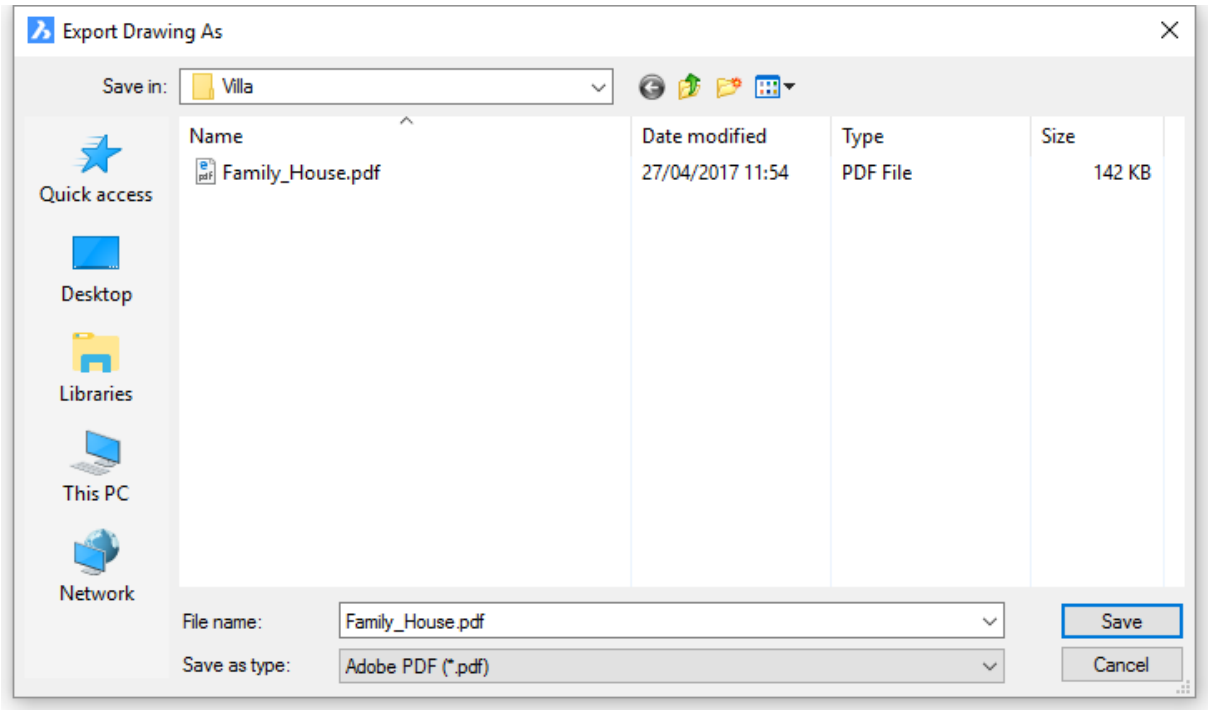
ExportPdf

Exports layouts in a drawing to a pdf docucment.

Accessing the Command

command bar: *exportpdf*
menu bar: *File| Export PDF...*

: ***exportpdf***
Displays a file dialog box:



Type a name in the *File name* field or accept the default name (*<DrawingName>.pdf*), then press the *Save* button.

Command Options

The options for PDF export can be found in the PDF export group in the Settings dialog or in the Pdf Options on the Publish dialog.

PDF export	
Pdf embedded fonts	☑ Enable embedded TTF fonts
Pdf TTF text as geometry	☐ Convert TTF text to geometry
Pdf SHX text as geometry	☐ Convert SHX text to geometry
Pdf simple geometry optimization	☑ Enable simple geometry optimization
Pdf zoom to extents mode	☑ Zoom to extents
Pdf Merge Control	[0] Lines Overwrite
Pdf layer support	[1] Use all layers with visible entities
Pdf layouts to export	[0] Active
Pdf papersize override	☐ Enable papersize override
Pdf overridden paper width	210
Pdf overridden paper height	297
Pdf use plotstyles	☑ Enables usage of plotstyles
Image anti-aliasing	☑ Enables anti-aliasing for images
Image compression	[1] JPEG
Vector Resolution DPI	2400
Image DPI	300

Option	Description
Pdf embedded fonts	Enables embedding of True Type fonts in the generated Pdf file.
Pdf TTF text as geometry	Enables the conversion of True Type font texts to geometry.
Pdf SHX text as geometry	Enables the conversion of SHX font texts to geometry.
Pdf simple geometry optimization	Enables simple geometry optimization: unify separate segments to one polyline; use of Bezier curve control points.
Pdf zoom to extents mode	When OFF use the scale and papersize of the page setup. When ON zoom the layout geometry to the papersize.
Pdf Merge Control	Controls the appearance of crossing lines. 0 - Lines Overwrite: The top most line is visible at the intersection. 1 - Lines Merge: The colors of crossing lines are blended into a new color.
Pdf layer support	Controls the use of layers in the generated Pdf file. 0 - Do not use layers. 1 - Only use visible layers. 2 - Use all layers, including OFF and FROZEN layers.
Pdf layouts to export	Controls which layouts are exported. 0 - Active layout only. 1 - All layouts in a single multi-sheet pdf file. 2 - Each layout in a separate single-sheet pdf file.
Pdf papersize override	Enables papersize override.
Pdf overridden paper width	Pdf width to be used in the papersize override in millimeters.
Pdf overridden paper height	Pdf height to be used in the papersize override in millimeters.
Pdf use plotstyles	Enables the usage of plotstyles.

Image anti-aliasing	Enables anti-aliasing for images that require upscaling when exporting.
Image compression	Sets the compression for exported images. 0 - No compression. 1 - JPEG compression.
Vector resolution DPI	Sets the resolution for vector graphics when exporting. Values between 72 and 40000 are accepted.
Image DPI	Sets the minimal resolution for raster images. The value cannot exceed the value of PdfVectorResolutionDPI.

Related Commands

Export - exports drawings in a variety of vector and raster formats.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.
Plot - plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.
Print - prints drawings to plotters and files through a dialog box.
-Publish - prints a saved sheet list (= a list of model space or paper space layouts).
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

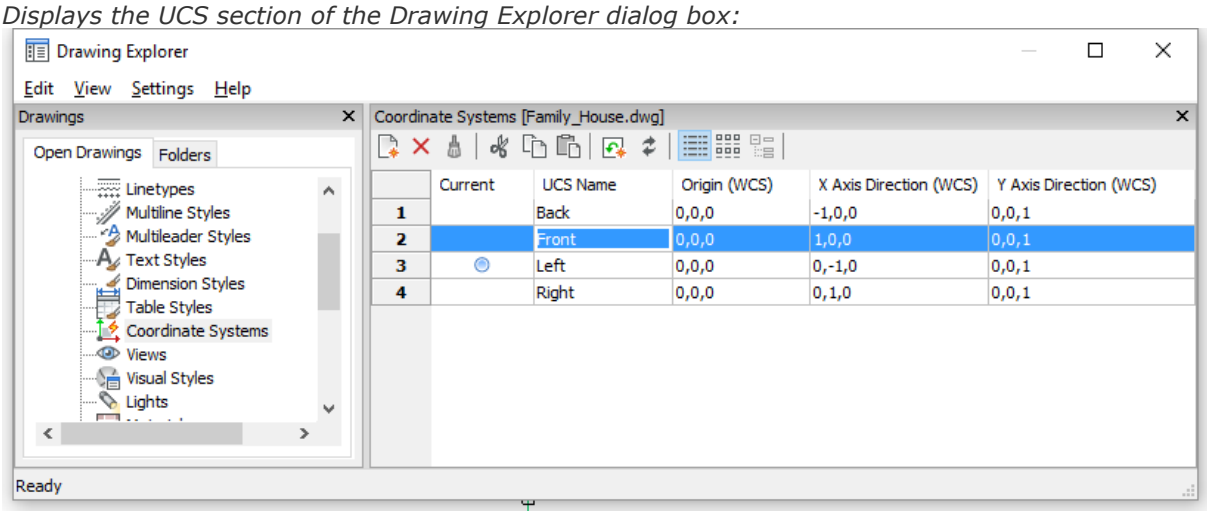
ExpUcs

Creates, modifies, and deletes named UCSs through the Drawing Explorer (short for "explore user-defined coordinate systems").

Accessing the Command

command bar: *expucs*
alias: *uc, dducs, explorer*
menu bar: *Tools | Drawing Explorer | Coordinate Systems*

: ***expucs***
Displays the UCS section of the Drawing Explorer dialog box:



*Create or modify named UCSs, and then click **X**.*
Press **F1** to access help.

Command Options

Option	Description
UCS Name	Lists the names of UCSs in the drawing; click to rename. UCS is short for "user-defined coordinate system."
Origin (WCS)	Reports the origin (0,0,0) of the UCS in x,y,z coordinates of the WCS; cannot be edited. WCS is short for "world coordinate system."
X Axis Direction (WCS)	Reports the direction of the x axis in WCS coordinates; cannot be edited.
Y Axis Direction (WCS)	Reports the direction of the y axis in WCS coordinates; cannot be edited.
Current	Indicates the current UCS.
Toolbar Icons	
New	Creates a new UCS; dismisses the Drawing Explorer, and then runs the Ucs command. See Ucs command.
Erase	Deletes the selected UCS; the current UCS cannot be erased.

Related Commands

Ucs - creates and modifies user-defined coordinate systems.
UcsIcon - controls the appearance of the UCS icon.
SetUcs - sets the UCS to a specified viewpoint.
Plan - changes the viewpoint to the plan view of the WCS or current UCS.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
Settings - sets parameters of UCSs.

Explorer - Opens the Drawing Explorer dialog box.

Extend

Extends open entities to a boundary.

Accessing the Command

command bar: *extend*
alias: *ex*
menu bar: *Modify | Extend*
toolbar: *Modify |*

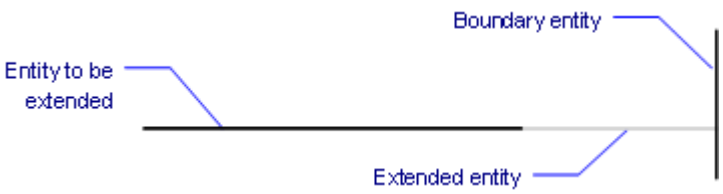
: ***extend***
Prompts you in the command bar:

Select boundary entities for extend <ENTER to select all>: (Choose one or more entities, or press Enter to choose all.)
Entities in set: 1

Select boundary entities for extend <ENTER to select all>: (Choose more entities, or press Enter to end entity selection.)

Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase]: (Choose the entity to extend, or enter an option or press shift, then pick an entity to trim.)

Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase]: (Choose another entity to extend, enter an option, or press Enter to exit the command or press shift, then pick an entity to trim.)



Command Options

Option	Description
Shift key	Trims the picked entities instead of extending them. When executing the Trim or the Extend command, pressing the shift key switches between trimming and extending.
Select boundary entities for extend	Chooses one or more entities as boundaries.
ENTER to select all	Press Enter to choose all entities in the drawing.
Select entity to extend	Chooses the entity to extend. When either end could be extended to a boundary, BricsCAD extends the end closest to the pick point.
Edge mode	Toggles edge mode; prompts you: **[Extend/No extend] <No extend>:** • Extend - extends entities to real and implied boundaries in 3D space. • No Extend - extends entities only to real boundaries.

Fence	Allows fence selection, an irregular selection window; prompts you: First point of fence - pick the first point of the fence. Next fence - pick the next point of the fence; press Enter to exit selection mode.
Crossing	Allows crossing rectangle selection; prompts you: First corner of crossing window - specify the first corner Second corner of crossing window - specify the second corner. The entities that cross the specified rectangle are extended.
Projection	Specifies the type of projection; prompts you: **Projection [No projection/xy plane of Ucs/current View] <UCS>:** • No projections - extend only those entities that intersect real boundaries in 3D space. • xy plane of UCS - project entities and boundaries to the xy plane of the current UCS, and then extends the projected entities that would intersect projected boundaries. • Current View - project the entities into the current view, and then extends them accordingly.
Erase	Deletes selected entities; prompts you: Select entities to delete - select the entities
Undo	Undoes the previous extend operation.
Shift-select to trim	Allows to switch to the Trim command. Press and hold the Shift-key to select the part of an entity to be trimmed at the intersection with the nearest cutting entities.

Procedures

To extend entities

Related Commands

Change - changes the length of lines.
Lengthen - changes the length of open entities.
Stretch - stretches entities longer and shorter.
Trim - trims entities to a cutting line.

Extension

Toggles extension entity snap.

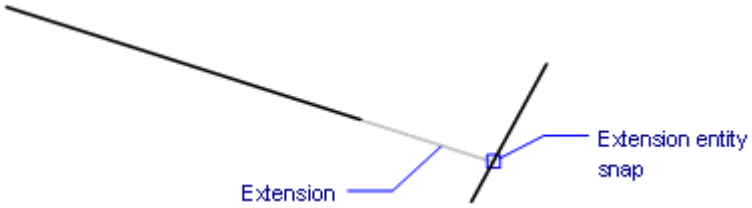
Accessing the Command

command bar: *extension*
transparent: *ext*
toolbar: *Entity Snaps* |

: ***extension***

No prompts in the command bar.

If extension entity snap is off, entering the command turns it on. And vice versa.



When extension entity snap is on, the cursor snaps to the intersection of one or two extended entities.

Command Options

Option	Description
Extension	Entering this command toggles extension entity snap. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

Extrude

Extrudes 2D entities to 3D solids.

Accessing the Command

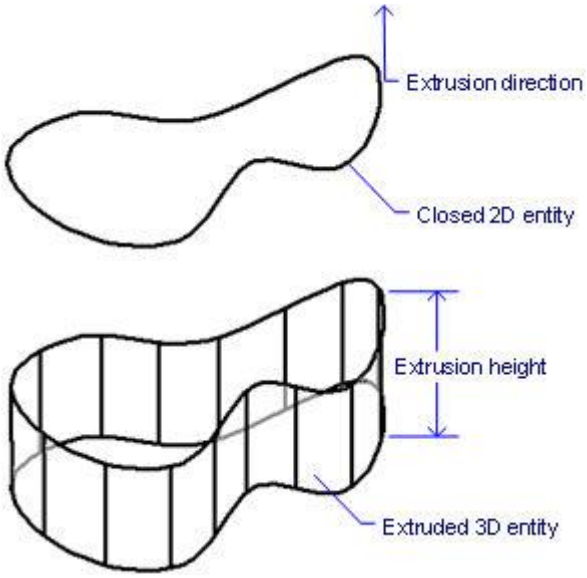
command bar: *extrude*
alias: *ext*
menu bar: *Model | 3D Solids | Extrude*
toolbar: *3d Solids |*

: ***extrude***
Prompts you in the command bar:

Select entities/subentities to extrude or set [MOde]: (Choose one or more closed 2D entities or faces of 3D solids.)

Entities in set: 1

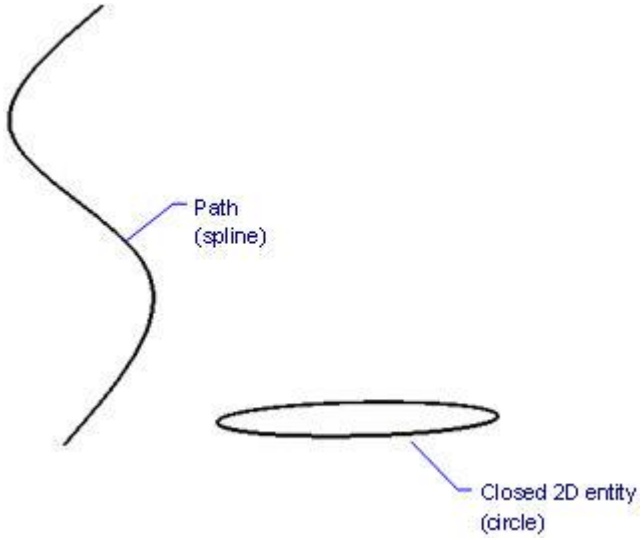
Select entities/subentities to extrude or set [MOde]: (Choose more entities, or press Enter to continue.)
Specify height of extrusion or [Direction/Path/Taper angle] <xx.xx>: (Enter a number, or pick two points, or press Enter to accept the default height, or choose an option.)

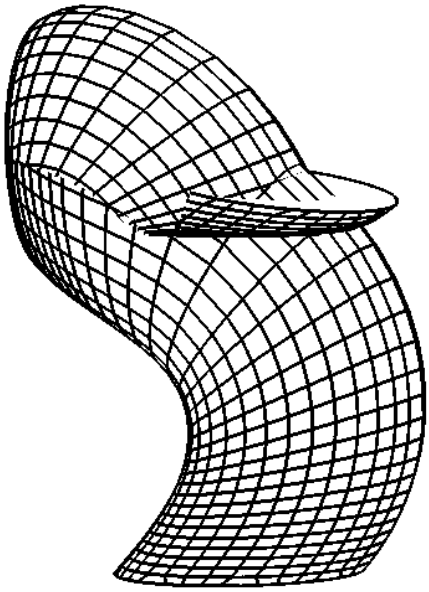
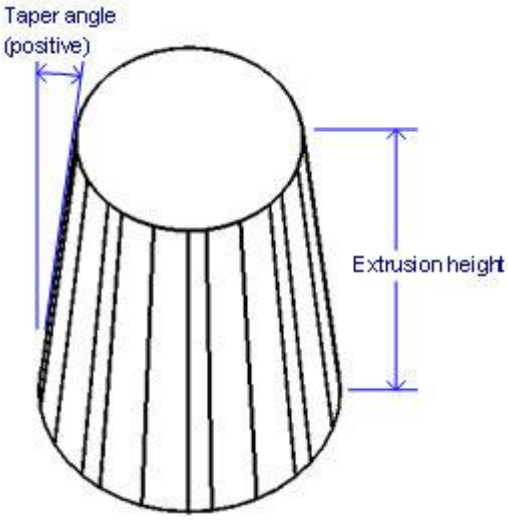


Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities.

Command Options

Option	Description
Select entities	Specifies the entities to be extruded. Choose one or more 2D closed entities, regions or faces of 3D solids. The resulting entity is either a 3D solid or a surface, depending on the current value of the Mode option:

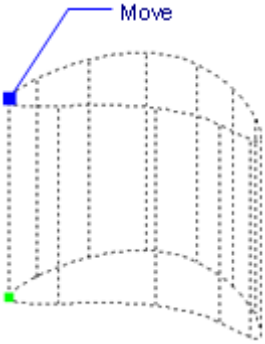
	Depending on the value of the DELOBJ system variable the source entities or either retained or deleted or you are prompted whether the entities are to be deleted or not..
Mode	Sets the entity type of the extrusion: solid or surface Prompts you: Choose type of created entity [SOlid/SUrface] <SOlid> - choose solid or surface or press Enter to accept the current mode.
Specify height of extrusion	Specifies the height of the extrusion. The extrusion height is measured perpendicularly to the selected entity. Use the Direction option to set the extrusion direction differently. • Enter a number - specifies the height; enter a negative number to extrude downwards. • Pick two points - the distance between two points specifies the height.
Direction	Sets the extrusion direction and height. Prompts you: Specify start point of direction - pick a point. Specify end point of direction - pick a point.
Path	Specifies the extrusion height and angle by another entity. Prompts you: Select extrusion path or [**Taper angle**] - Pick an open entity, such as a line, arc, spline, or polyline.  The path entity cannot lie in the same plane as the extrusion entity.

	
Taper Angle	Specifies the taper angle for the extrusion. Prompts you: Specify angle of taper for extrusion <0>: • Enter a number - specifies the angle; enter a negative number to taper outwards. • Define the angle by picking two points. The angle must be shallow enough not to cut off that the top of the extrusion. 

Grips Editing

Extrusion can be edited directly through grips:

1. Select the extruded solid. Notice that it has two grips.
2. Drag a grip to move the extrusion. (Both grips perform the same functions: move.)



Related Commands

dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
Loft - creates 3D solids passing through two or more cross sections.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.

Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.

F

Field

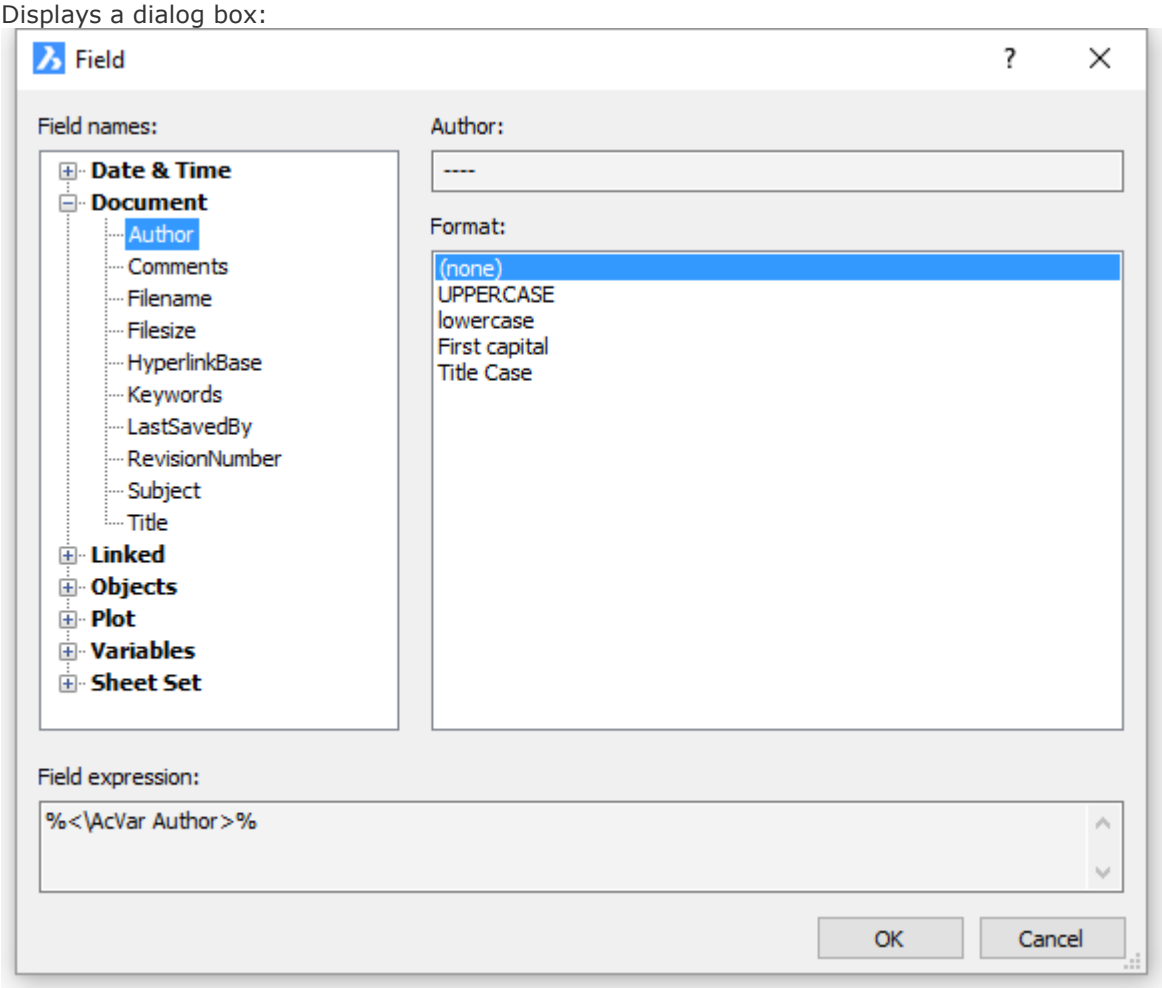
Inserts a field in a multiline text, table cell or attribute.
A field is variable text to display data such as drawing properties (filename, save date, ...), entity properties (Length, Area, Layer, ...) user defined properties, plot settings and variables.
If no value is available, hyphens display (----)

Accessing the Command

command bar: *field*
alias: none
menu bar: *Insert| Insert Field*
toolbar: *Insert |*

: ***field***

Displays a dialog box:



Choose field, then click OK.

Prompts you in the command bar:
Specify start point or [Height/Justify]

Command Options

Option	Description
Height	Specifies the text height. Prompts you: **Specify height <current height>**
Justify	Sets the text justification. Prompts you: **Enter justification [TL/TC/TR/ML/MC/MR/BL/BC/BR]<Top Left>**

Procedures

General procedure to create fields
To display an entity property in a field
To edit a field
To update a field

Related Commands

AttDef - defines how a block's attributes will be created through a dialog box.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Dwgprops - opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.
Mtext and -MText - places paragraph text.
Updatefield - updates fields in the selected Mtext entities.
Table - creates a table entity in the drawing.

FileOpen

Opens drawing (DWG), template (DWT), and interchange (DXF) files from the command line.

Accessing the Command

command bar: *fileopen*

: ***fileopen***
Prompts you in the command bar:
Drawing name: (Type the name of a drawing, including the path.)
If the file extension is not specified, BricsCAD searches for a drawing (DWG) file.

Command Options

No options.

Related Commands

Open - opens drawing (DWG), template (DWT), and interchange (DXF) files.

Files

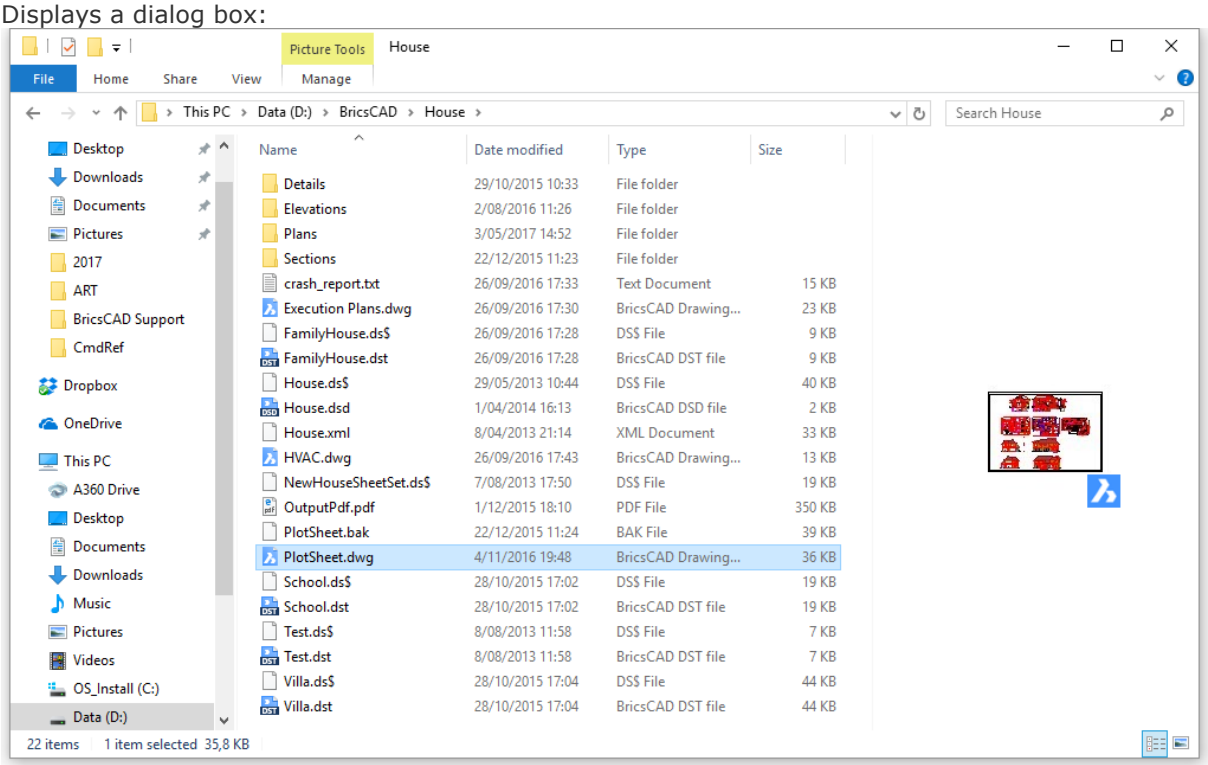
Opens Windows Explorer for accessing files.

Accessing the Command

command bar: *files*
transparent: *'files*

: ***files***

Displays a dialog box:



Command Options

See Windows documentation.

Related Commands

Cal - runs the Calculator application.
Start - starts other applications.
Shell - opens the operating system command prompt window; runs other applications.

Fill

Toggles the display of filled and hatched areas, such as wide polylines and 2D solids.

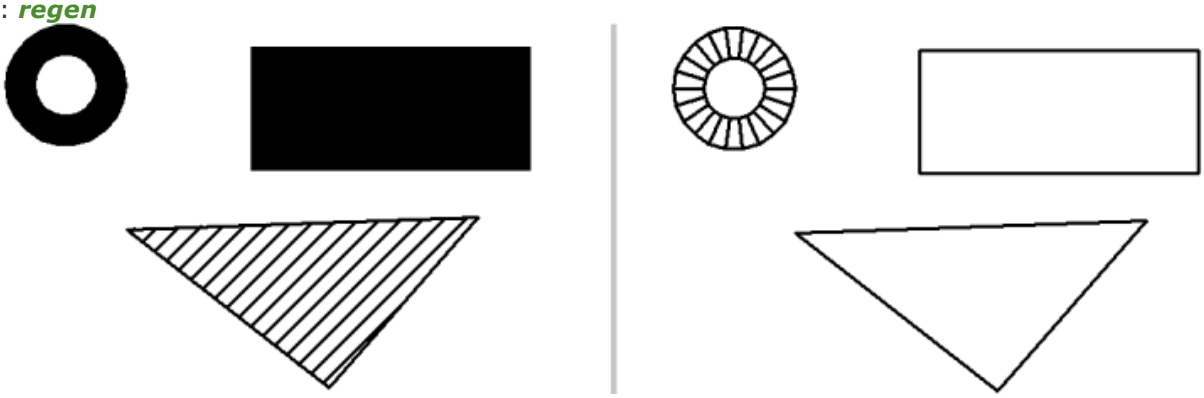
Accessing the Command

command bar: *fill*
transparent: *'fill*

: ***fill***

Prompts you in the command bar:
FILLMODE is currently on [ON/OFf/Toggle] <On>: *(Enter an option.)*

*To come into effect, this command must be followed by the **Regen** command:*
: ***regen***



Donut, rectangle, and hatched area. Left: Fill on; right: Fill off.

Command Options

Option	Description
OFF	Turns off the display of filled areas.
Toggle	Toggles fill mode; if on, turns it off, and vice versa.
On	Turns on the display of filled areas.

Related Commands

Settings - changes the FillMode system variable through a dialog box.

This setting affects the entities drawn by the following commands:
PLine - draws polylines with width.
Donut - draws fat circles out of polylines.
Polygon - draws regular polygons out of polylines.
Rectang - draws rectangles out of polylines.
Solid - draws solid-filled entities with 3 and 4 sides.
Trace - draws wide lines.

Fillet

Fillets intersections; joins intersecting lines with an arc of any suitable radius.

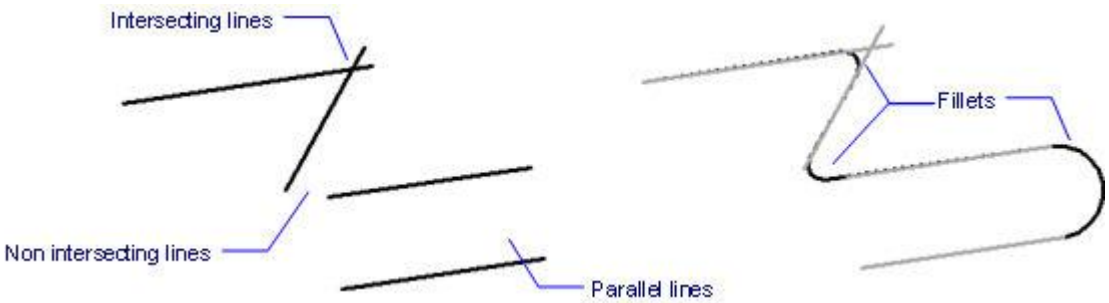
Accessing the Command

command bar: *fillet*
alias: *f*
menu bar: *Modify | Fillet*
toolbar: *Modify |*

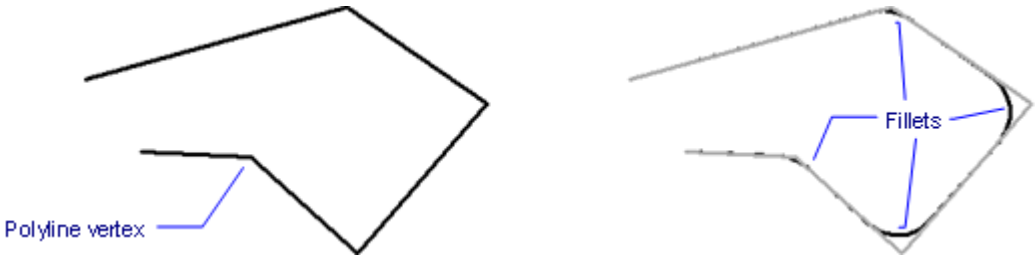
: ***fillet***
Prompts you in the command bar:

Fillet (radius=10): Select first entity or [Settings/Polyline/Radius/Trim/Undo/Multiple]: (Choose an entity, enter an option, or enter R to change the radius.)

Select second entity (select with pressed SHIFT to make corner): (Choose another entity.)



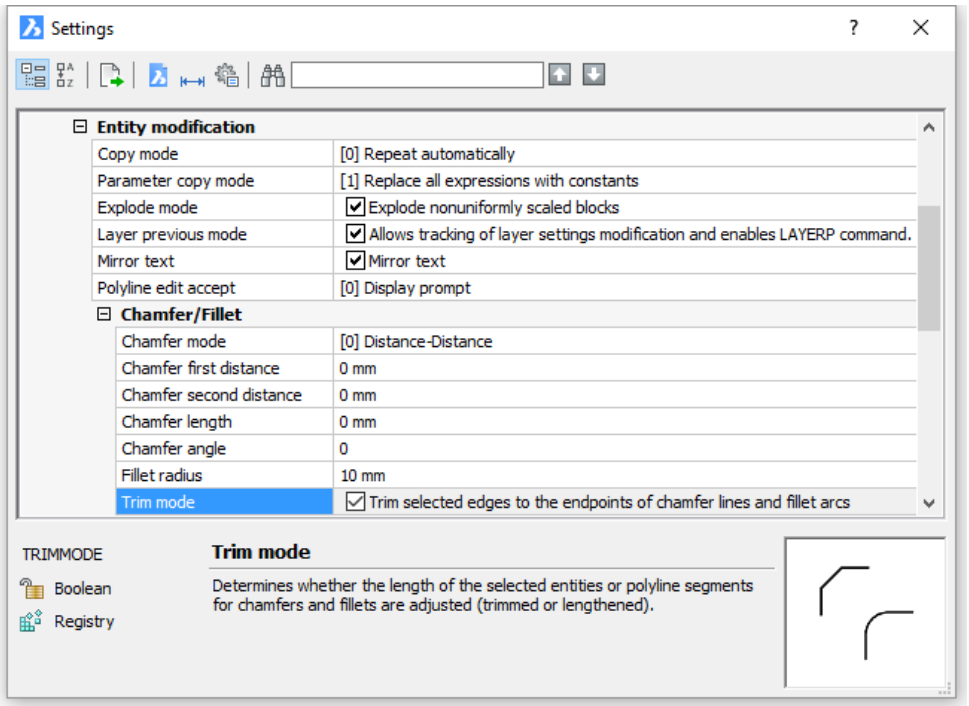
Fillets can be applied to intersecting, non-intersecting, and parallel lines.



Fillets are applied to all vertices of a polyline.

Command Options

Option	Description
Select first entity	Choose an entity, or enter R to change the radius. The following entities can be filleted: • Lines, including parallel lines. • All vertices of a single 2D polyline; two polylines cannot be filleted. • Rays, including parallel rays. • Infinite lines, including parallel infinite lines. • Arcs. • Circles.

Select second entity	Choose the second entity. Press and hold the Shift key to create a corner connection. 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 entities are not of equal length, the second entity is extended or trimmed if Trim Mode = Trim.
Settings	Displays the Chamfer/Fillet section of the Settings dialog box: • Fillet radius - sets the radius of the fillet arc. • Trim mode - toggles whether the entities are trimmed back to the fillet radius. 
Polyline	Fillets the all vertices of a polyline. Prompts you: Select 2D polyline to fillet - choose a single polyline.
Radius	Prompts you to type the Fillet radius in the command bar.
Trim	Sets the trim mode for chamfers and fillets. Prompts you: [Trim/No] <Trim> - choose an option or press Enter to keep the current mode. • Trim: Trims or extends the selected entities. • No: Creates the chamfer or fillet, but leaves the selected entities unchanged.
Undo	Undoes the last fillet when in Multiple mode.
Multiple	Lets you create multiple fillets using the same settings.

Procedures

Filleting two entities or polyline segments

Filleting all vertices of a polyline
Filleting two parallel lines

Related Commands

Break - cuts parts out of entities.
Chamfer - chamfers intersections.
dmChamfer - creates an equal distance chamfer between adjacent faces sharing a sharp edge.
dmFillet - creates a smooth fillet between adjacent faces sharing a sharp edge.
Extend - extends open entities to a boundary.
Settings - specifies parameters for fillets.
Trim - trims entities where they cross other entities.

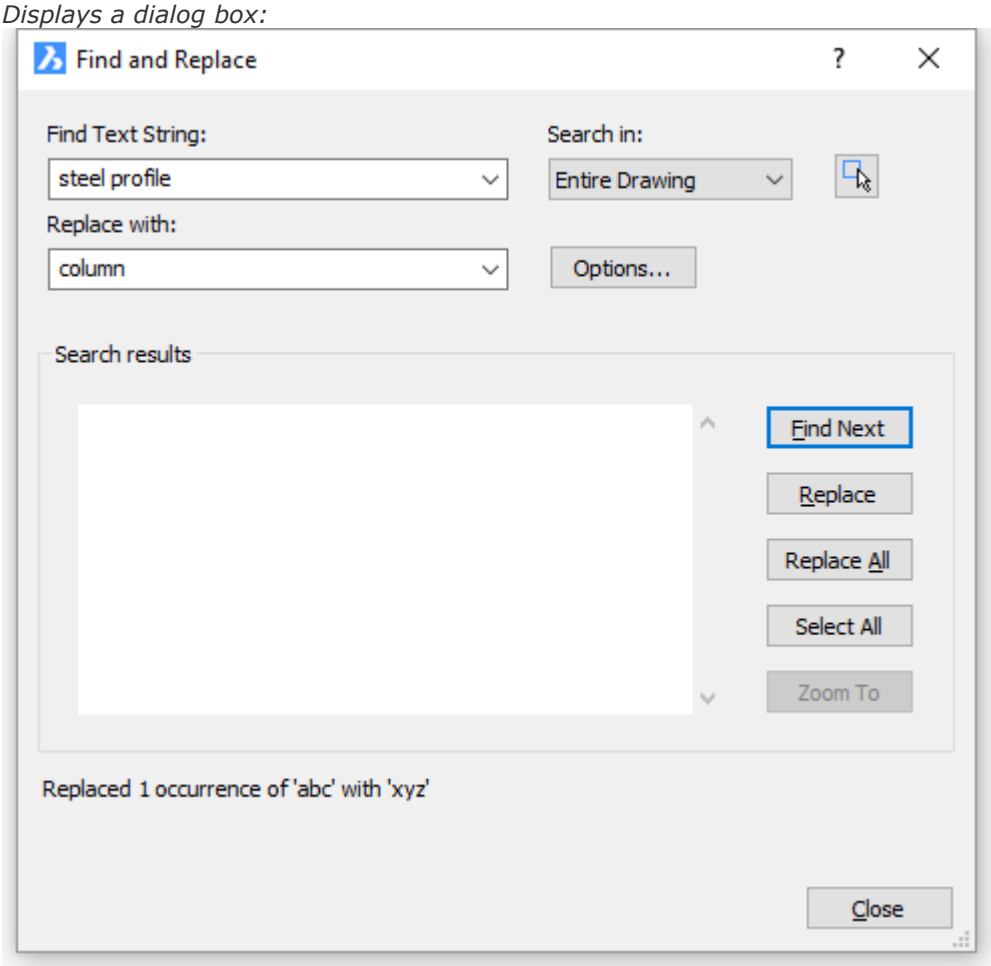
Find

Searches for text and optionally replaces it; searches through text blocks, attributes, dimensions, and hyperlinks.

Accessing the Command

command bar: *Find*
menu bar: *Edit | Find*

: ***find***
Displays a dialog box:

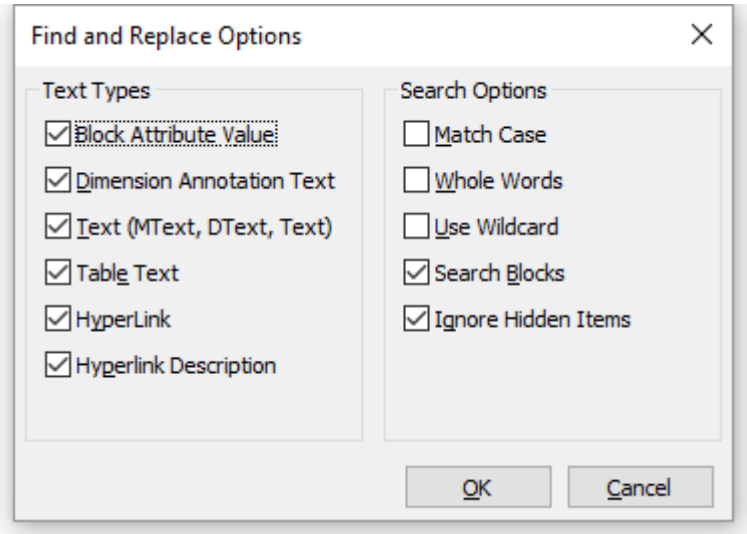


*Enter text to look for or replace, and then click **Find Next**.*

Command Options

Option	Shortcut	Description
Find Text String		Specifies the text to be located.
Replace With		Specifies the text with which to replace the located text. Leave this field empty when only searching for text.
Search In		Specifies the range of search: • Entire Drawing - searches all text in the drawing.

		• Current Selection - searches only among the selected entities.
Select Entities		Dismisses the dialog box so that you can select the entities to search; prompts you: Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to return to the dialog box.
Options		Displays the *Find and Replace Options* dialog box; allows you to specify the kinds of text entities to search.
Find Next	Alt+F	Finds the next occurrance of the text string.
Replace	Alt+R	Replaces the found text string with the replacement string.
Replace All	Alt+A	Replaces all occurances of the text string with the replacement string.
Select All		Selects all occurances of the text string.
Zoom To		Zooms to the part of the drawing containing the found text string.



Choose options, and then click ***OK***.

Find and Replace Options	**Description**
Block Attribute Value	Searches the values of block attributes as well.
Dimension Annotation Text	Searches dimension text as well.
Text (MText, DText, Text)	Searches regular text as well.
Table Text	Searches texts in tables.
Hyperlink	Searches hyperlinks (URLs) as well.
Hyperlink Description	Searches hyperlink descriptions as well.
Match Case	Searches the case of the text:

	• Yes - finds text that matches upper and lower case. For example, searching for "BricsCAD" matches BricsCAD but not bricscad. • No - finds all text, regardless of upper or lowercase.

Whole Words Only	Searches for full words: • Yes - finds only text that matches entire words. For example, searching for "CAD" matches CAD but not BricsCAD. • No - finds all text, including text embedded in longer words.
Use Wildcard	Allows to use wildcard characters such as * and ?
Search blocks	Searches for matching text strings inside blocs.
Ignore Hidden Items	If checked, ignores content on hidden and frozen layers.

Procedures

To set the Find and Replace options
To find text
To replace text

Related Commands

Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Hyperlink - adds hyperlinks to entities.
Spell - checks the spelling of words.
Properties - modifies the properties of text.

Flatshot

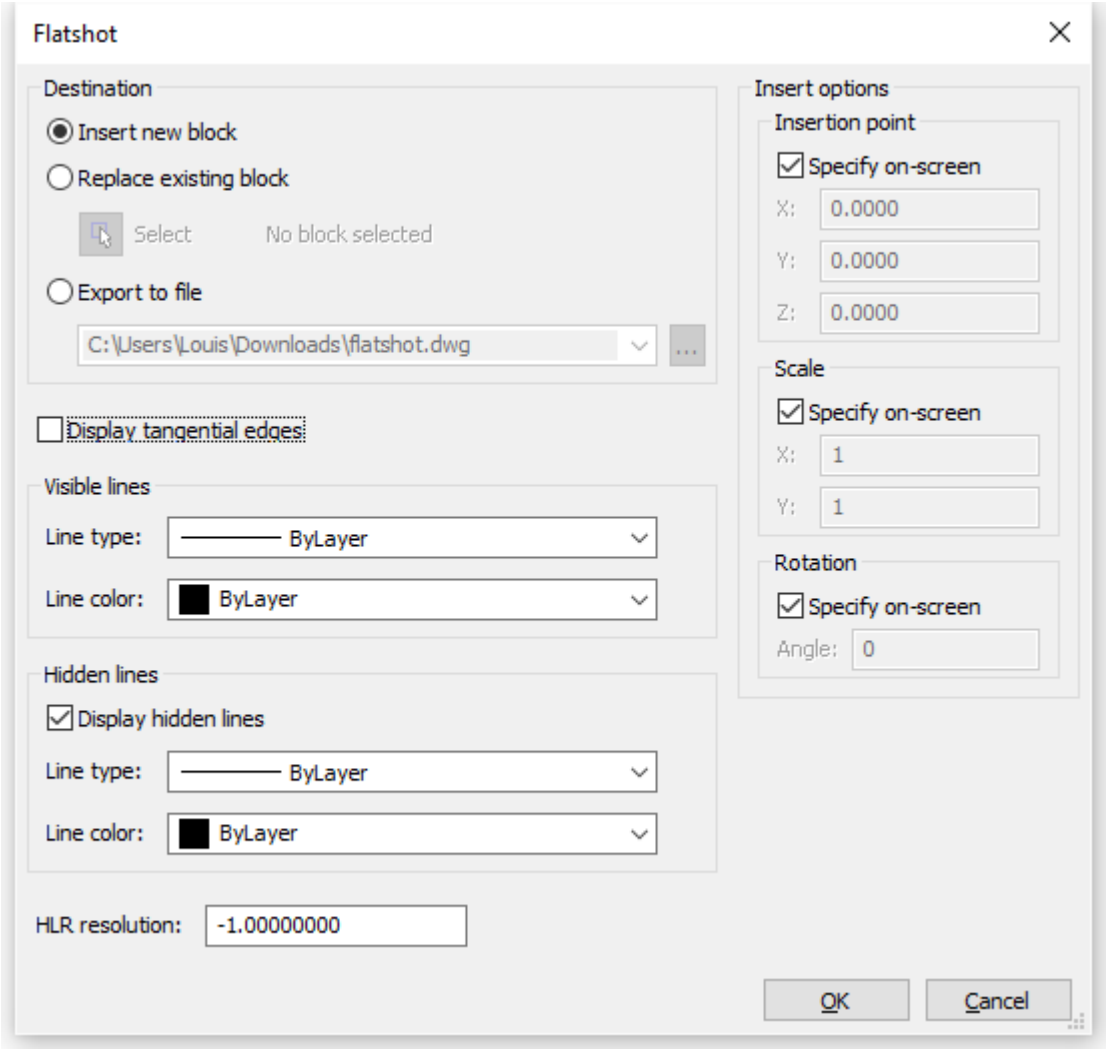
Creates a hidden line representation of all 3D solids in model space as a block or a new drawing.

Accessing the Command

command bar: *flatshot*
menu bar: *Model | Flatshot*
toolbar: *Section |*

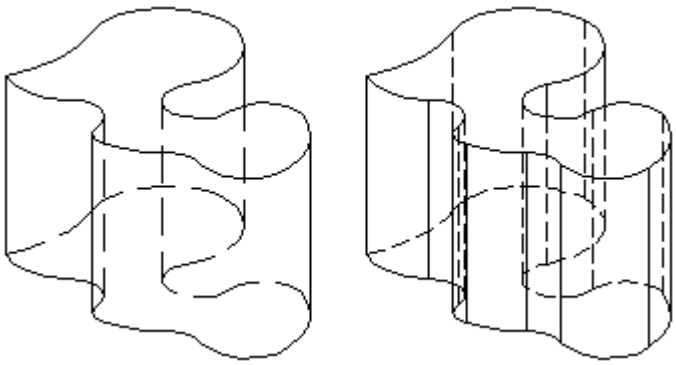
: ***flatshot***

Displays a dialog box:



Command Options

Option	Description
Output new block record	Defines the type of the hidden line representation: block or file
Create new block record	Creates a new block in the current drawing.

Replace existing block record	Redefines an existing block. If no block is selected, an alert displays: No block record selected To select a block: 1. Click the *Select block* button ( ). The *Flatshot* dialog box closes temporarily. You are prompted to select an existing block. 2. Click the block in the drawing. The *Flatshot* dialog box reopens. The name of the selected block shows on the *Flatshot* dialog box.
Export to file	Exports the hidden line representation to a file. Click the *Browse* button to locate and name the export file on the *Select export file* dialog box:
Display tangential edges	A tangential edge is the transition line between two tangent faces. It is an imaginary line between the two faces. In most cases tangential edges need to be removed.  *Tangential edges removed (left) - tangential edges displayed (right).*
Visible lines	Sets the linetype and color for the visible lines.
Hidden lines	Sets the linetype and color for the hidden lines.
Insert options	Defines the insert options for a new block
Position on screen	Lets you define the insertion point of the block on screen.
X: 0.0 Y: 0.0 Z: 0.0	Type the coordinates of the insertion point in the *X*, *Y* and *Z* fields.
Scale on screen	Lets you define the scaling of the block on screen.
X: 1.0 Y: 1.0	Type the X- and Y-scaling factor in the *X* and *Y* fields.

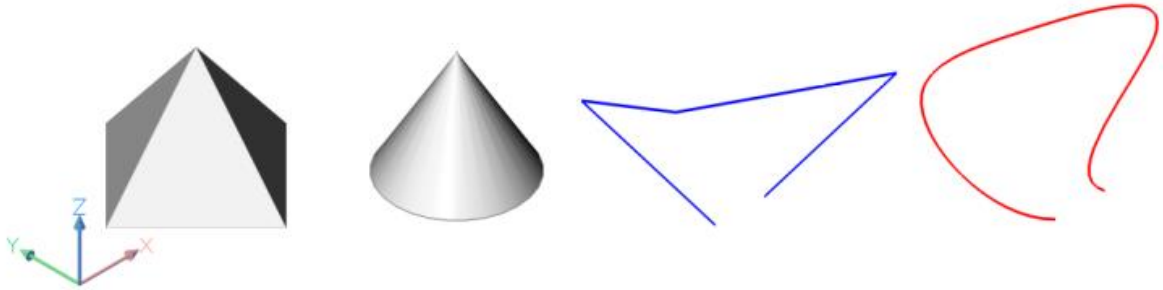
Rotation on screen	Lets you define the rotation angle of the block on screen.
Angle: 0.0	Type the rotation angle in the *Angle* field.
HLR Resolution	Sets the *AcisHlrResolution* user preference. Specifies the accuracy to be used for mass properties calculations. This accuracy is relative: the accuracy of the result depends on the magnitude of the calculated value. Acceptable value range: 0.01 - 0.000000000001 (1.e-2 to 1.e-12). Specifies the smallest distance taken into consideration during Hidden Line Removal calculation. A negative value means that auto-calibration based on the size of the model will be used. This is highly recommended. When dealing with objects that have very small sizes the value can be set to 0.001 or smaller. The value of the *AcisHlrResolution* user preference can also be set in the SETTINGS dialog, under *Program Options / Modeler / Modeler Properties for full ACIS*.
OK	Creates the hidden line representation.

Related Commands

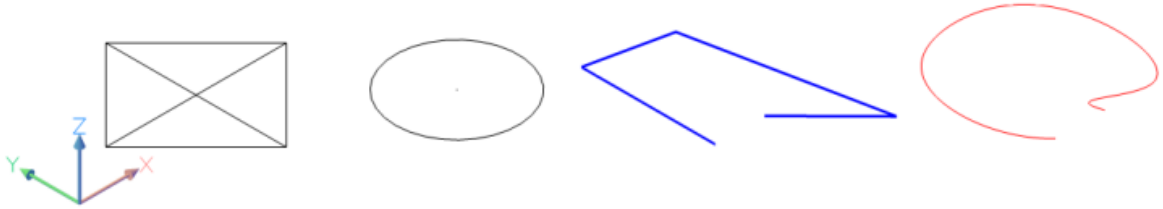
Hide - removes hidden lines from 3D entities.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
Shademode - sets the shading style for 3D drawings.
ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drafting views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewSection - creates a cross section view based on a standard drafting view generated by the VIEWBASE command in a paper space layout.

Flatten

Flattens 2D and 3D entities: projects the entities onto the XY-plane of the current view. Although the command can be executed in any view orientation, the result makes sense in a top view only.



From left to right: pyramid, cone, 3D polyline, spline



Isometric view of the entities after executing the Flatten command in top view.

The pyramid is converted to a square with its diagonals; the cone becomes a circle, with a point in the center; the 3D polyline and the spline are converted to a 2D polyline.

Accessing the Command

command bar: *flatten*
menu bar: *Modify | Flatten*
toolbar: *Modify |*

: ***flatten***
Prompts you in the command bar:

Select entities [new ucs ELevation/allow EXplode]: (Choose one or more entities, or choose an option.)

Entities in set: 1

Select entities [new ucs ELevation/allow EXplode]: (Choose more entities, or press Enter to end entity selection.)

The selected entities are flattened.

Command Options

Option	Description
Select entities to flatten	Chooses the entities to be flattened. The entities will be projected onto the XY plane of the current view. In top view, the entities will be projected onto the XY-plane of the current coordinate system (WCX or UCS).
New UCS elevation	Specifies the new elevation for the 2D polylines in the selection set. The elevation and thickness can be changed with the Elev command.
Explode	Allows compound entities, such as 3D solids, to be exploded.

Related Commands

Elev - sets the thickness and elevation of entities.
FlatShot - creates a hidden line representation of all 3D solids in model space as a block or a new drawing.
Section - creates 2D region slices of 3D solid models.
SolidEdit - creates 2D entities from 3D solid models.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.

From

Allows to enter relative coordinates with respect to a reference point whenever a command prompts to enter a point (transparent command).

Accessing the Command

transparent: *from*
toolbar: *Entity Snaps* |

<running command prompt>: ***from***
Prompts you in the command bar:

Base point: **(Identify the reference point)**

Offset or regular point: **(Enter relative coordinates: @ x,y,z)**

Command Options

None.

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

G

Geometric Constraints

DelConstraint

Removes all dimensional and geometrical constraints from an entity.

Accessing the Command

command bar: *delconstraint*
menu bar: *Parametric | Delete 2D Constraints*
toolbar: *2D Constraints |* ✕

: ***delconstraint***
Prompts you in the command bar:

Select entities to delete all constraints: **(select an entity or entities)**
Entities in set: 1

Select entities to delete all constraints: **(select more entities or press Enter to delete all constraints from the selection set)**

The dimensional and geometric constraints of the selected entities are deleted.

Command Options

No options

Related Commands

-Parameters - manages 2D dimensional constraint parameters and user variables in the current drawing.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

ConstraintBar - controls the display of the constraint bar of entities.
DcDisplay - allows to set dimensional constraints to be displayed or hidden.

GeomConstraint

Applies geometric relationships between entities and/or valid constraint points on entities.

Valid constraint points on an entity:

Entity Type	Valid Constraint Points
Line	Endpoints, Midpoint
Arc, Elliptical Arc	Endpoints, Center, Midpoint
Circle, Ellipse	Center
Polyline	Endpoints, vertices, midpoints of line and arc segments, center of arc segments
Spline	Endpoints
Insert (Block, Xref, Text, Mtext, Attribute, Table)	Insertion point

Valid constraint entities

Entity Type
Line
Polyline Segment
Circle
Arc, Elliptical Arc
Ellipse
Spline

Accessing the Command

command bar: *geomconstraint*
menu bar: *Parametric | Geometric Constraints| item*

toolbar: *Geometric Constraints |*

: ***geomconstraint***
Prompts you in the command bar:
Enter constraint type [Horizontal / Vertical / Perpendicular / PArallel / Tangent / SMooth / Coincident / CONcentric / COLlinear / Symmetric / Equal / Fix] <Coincident>: (Enter an option).

Command Options

Option	Description
Horizontal	Constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. This option is equivalent to the GcHorizontal command ( ).
Vertical	Constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. This option is equivalent to the GcVertical command ( ).
Perpendicular	Constrains two lines to lie perpendicularly to each other.

	This option is equivalent to the GcPerpendicular command ( ).

Parallel	Forces two lines to be parallel to each other. This option is equivalent to the GcParallel command ( ).
Tangent	Constrains two entities to maintain a point of tangency to each other or their extensions. This option is equivalent to the GcTangent command ( ).
Smooth	Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. This option is equivalent to the GcSmooth command ( ).
Coincident	Applies a coincident geometrical constraint to two points or constrains a point to an entity. This option is equivalent to the GcCoincident command ( ).
Concentric	Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. This option is equivalent to the GcConcentric command ( ).
Collinear	Forces lines to be collinear. This option is equivalent to the GcCollinear command ( ).
Symmetric	Constrains two entities or points to lie symmetrically with respect to a selected line. This option is equivalent to the GcSymmetric command ( ).
Equal	Constrains arcs and circles to the same radius, or lines to the same length. This option is equivalent to the GcEqual command ( ).
Fix	Constrains points and entities at a fixed position. This option is equivalent to the GcFix command ( ).

Related Commands

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident

Applies a coincident geometrical constraint to two points or constrains a point to an entity.
This command is equivalent to the Coincident option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gccoincident*
menu bar: *Parametric | 2D Constraints | Coincident*
toolbar: *2D Constraints |*

: ***gccoincident***
Prompts you in the command bar:

Select first point or [Entity/Autoconstrain] <Entity>: (select a point on an entity or press Enter, then select an entity.)

Select point or [Multiple]: (select a point or type M to select multiple points)

Select second point or [Entity] <Entity>: (select a point on an entity or press Enter, then select an entity.)

Command Options

Option	Description
Select first point	Snap to a valid constraint point on an entity.
Select second point	Snap to a valid constraint point on an entity. The second point is moved to coincide with the first point.
Entity	Select a valid constraint entity.
Select point	Snap to a valid constraint point on an entity. The selected point is placed on the selected entity.
Multiple	Subsequently snap to a valid constraint point on multiple entities. Right click to stop selecting points. The selected points are placed on the selected entity.
Select first point	Snap to a valid constraint point on an entity.
Entity	Select a valid constraint entity. The selected entity is moved to the selected point.
Autoconstrain	Select multiple entities. Coincident constraints are applied to coincident unconstraint points of the entities in the selection set. The number of constraints applied displays in the command bar.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcCollinear

Forces lines or polyline segments to be collinear.
This command is equivalent to the Collinear option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gccollinear*
menu bar: *Parametric | 2D Constraints | Collinear*
toolbar: *2D Constraints |*

: ***gccollinear***
Prompts you in the command bar:

Select first entity or [Multiple]: (select a line or polyline segment.)

Select second entity: (select a line or polyline segment.)
The second entity is made collinear to the first.

Command Options

Option	Description
Multiple	
Select first entity	Select a line or polyline segment.
Select entity to make collinear to first:	Select multiple lines or polyline segments (segments of the same polyline accepted). Right click to stop selecting entities.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcConcentric

Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
This command is equivalent to the Concentric option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcconcentric*
menu bar: *Parametric | 2D Constraints | Concentric*
toolbar: *2D Constraints |*

: ***gcconcentric***
Prompts you in the command bar:

Select first entity: (select a circle, arc, ellipse or elliptical arc.)

Select second entity: *(select a circle, arc, ellipse or elliptical arc.)*

The second entity is moved to be concentric with the first.

Command Options

No options.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcEqual

Constrains arcs and circles to the same radius, or lines and polyline segments to the same length. This command is equivalent to the Equal option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcequal*
menu bar: *Parametric | 2D Constraints | Equal*
toolbar: *2D Constraints |* =

: ***gcequal***
Prompts you in the command bar:

Select first entity or [Multiple]: (Select a line, arc, circle or polyline segment.)

Select second entity: (Select a second entity of the same kind (straight or arched) as the first entity.)

The dimensions of the two entities are constrained: lines an linear polyline segments are resized to the same length; circles, arcs and arc polyline segments are resized to the same radius.

Command Options

Option	Description
Multiple	
Select first entity	Select a line, arc, circle or polyline segment.
Select second entity	Select multiple entities to be made equal to the first entity. All selected entities must be of the same kind (straight or arched). Right click to stop selecting entities. Segments of the same polyline are accepted.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcFix

Constrains points and entities at a fixed position.
This command is equivalent to the Fix option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcfix*
menu bar: *Parametric | 2D Constraints | Fix*
toolbar: *2D Constraints |*

: ***gcfix***
Prompts you in the command bar:
Select point or [Entity]: (*Select a valid constrain point in the drawing*)

Command Options

Option	Description
Select point	Select a valid constraint point on an entity. The selected point cannot be moved anymore.
Entity	Select a valid constraint entity. The position of the entity cannot be modified anymore. You can modify the length of a line or polyline segment in either direction though.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcHorizontal

Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system.
This command is equivalent to the Horizontal option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gchorizontal*
menu bar: *Parametric | 2D Constraints | Horizontal*
toolbar: *2D Constraints |*

: ***gchorizontal***
Prompts you in the command bar:
Select an entity or [2Points] <2Points>: (*Select a line or linear polyline segment or press Enter, then select two points.)*

Command Options

Option	Description
Entity	The selected line or linear polyline segment is forced to be parallel to the X-axis of the current coordinate system.
2Points	Select two valid constraint points on the same entity or on different entities. The two selected points are forced to lie on a line which is parallel to the X-axis of the current coordinate system.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcParallel

Forces two lines or linear polyline segments to be parallel to each other.
This command is equivalent to the Parallel option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcparallel*
menu bar: *Parametric | 2D Constraints | Parallel*
toolbar: *2D Constraints |*

: ***gcparallel***
Prompts you in the command bar:

Select first entity: (select a line or linear polyline segment.)

Select second entity: (select a line or linear polyline segment.)
The second entity is made parallel with the first.

Command Options

No options.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcPerpendicular

Constrains two lines or polyline segments to lie perpendicularly to each other.
This command is equivalent to the Perpendicular option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcperpendicular*
menu bar: *Parametric | 2D Constraints | Perpendicular*
toolbar: *2D Constraints |*

: ***gcperpendicular***

Prompts you in the command bar:

Select first entity: (select a line or linear polyline segment.)

Select second entity: (select a line or linear polyline segment.)
The second entity is made perpendicular to the first.
Segments of the same polyline are accepted.

Command Options

No options.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcSmooth

Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. This command is equivalent to the Smooth option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcsmooth*
menu bar: *Parametric | 2D Constraints | Smooth*
toolbar: *2D Constraints |*

: ***gcsmooth***
Prompts you in the command bar:

Select first entity: (Select a spline.)

Select second entity: (Select a spline, line, arc or polyline.)

The spline is updated to be contiguous with the secondly selected entity.

Command Options

No options.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcSymmetric

Constrains two entities or points to lie symmetrically with respect to a selected line.
This command is equivalent to the Symmetric option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcsymmetric*
menu bar: *Parametric | 2D Constraints | Symmetric*
toolbar: *2D Constraints |*

: ***gcsymmetric***
Prompts you in the command bar:

Select first entity or [2Points] <2Points>: (Select the first entity, or press Enter, then select two points.)

Select second entity: (Select the second entity.)

Select symmetry line: (Select a line.)

Command Options

Option	Description
Select first entity	Select a valid constraint entity.
Select second entity	Select a valid constraint entity.
Select symmetry line	Select a line. The second entity is placed symmetrically to the first entity with respect to the selected line. Lines: The angle with the symmetry line is made symmetric, not the position of the endpoints. Circles: The center and radius are made symmetric. Arcs: The center and radius are made symmetric, not the endpoints. Ellipses: The center, major and minor axes are made symmetric. Elliptical arcs: The center, major and minor axes are made symmetric, not the endpoints.
2Points	
Select first point	Select a valid constraint point.
Select second point	Select a valid constraint point.
Select symmetry line	Select a line. The second point is moved to lie symmetrically to the first point with respect to the selected line.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcTangent

Constrains two entities to maintain a point of tangency to each other or their extensions.
This command is equivalent to the Tangent option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gctangent*
menu bar: *Parametric | 2D Constraints | Tangent*
toolbar: *2D Constraints |*

: ***gctangent***
Prompts you in the command bar:

Select first entity: (Select a line, polyline segment, circle, arc, ellipse or elliptical arc.)

Select second entity: (Select a circle, arc, ellipse or elliptical arc; or a line or linear polyline segment, if the first object was not a line or polyline segment.)
The selected entities are placed tangent to each other.

Entities can be tangent even if they do not touch each other physically.

Command Options

No options.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcVertical - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

GcVertical

Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system.
This command is equivalent to the Vertical option of the GEOMCONSTRAINT command.

Accessing the Command

command bar: *gcvertical*
menu bar: *Parametric | 2D Constraints | Vertical*
toolbar: *2D Constraints |* ||

: ***gcvertical***
Prompts you in the command bar:
Select an entity or [2Points] <2Points>: (*Select a line or linear polyline segment or press Enter, then select two points.)*

Command Options

Option	Description
Entity	The selected line or linear polyline segment is forced to be parallel to the Y-axis of the current coordinate system.
2Points	Select two valid constraint points on the same entity or on different entities. The two selected points are forced to lie on a line which is parallel to the Y-axis of the current coordinate system.

Related Commands

GeomConstraint - applies geometric relationships between entities and/or points on entities.
Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

GcCoincident - applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcCollinear - forces lines to be collinear.
GcConcentric - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcEqual - constrains arcs and circles to the same radius, or lines to the same length.
GcFix - constrains points and entities at a fixed position.
GcHorizontal - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcParallel - forces two lines to be parallel to each other.
GcPerpendicular - constrains two lines to lie perpendicularly to each other.
GcSmooth - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric - constrains two entities or points to lie symmetrically with respect to a selected line.
GcTangent - constrains two entities to maintain a point of tangency to each other or their extensions.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

Gcenter

Toggles Geometric Center entity snap; snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.

Accessing the Command

command bar: *gcenter*
transparent: *gce*
toolbar: *Entity Snaps* |

: ***GCenterNTER***
Displays no prompts in the command bar.

Command Options

Option	Description
Gcenter	Entering the command toggles Geometric Center entity snap. It snaps to the centroid of any closed entities and planar faces of 3D solids. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.
Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.

GenerateBoundary

Creates closed polylines from faces of 3D solids as well as from the boundaries detected when the Enable Boundary Detection option of SELECTIONMODES is activated.

Accessing the Command

command bar: *generateboundary*
menu bar: *Draw | Boundaries | Generate Boundary*
toolbar: *Draw | Boundaries |*
Quad: *2D Editing |*

: ***generateboundary***
Prompts you in the command bar:
Select a planar enclosed point, a planar face, or a hatch to generate a boundary: *(click inside closed regions, select faces of 3D solids or hatches, then right click.)*

Generated boundaries for X object(s)

Spline or elliptical segments of boundaries are converted to multi-segmented polyline edges.

Command Options

No command options.

Related Commands

Boundary - creates a polyline that forms a boundary inside a closed area through a dialog box.
-Boundary - creates a polyline boundary inside closed areas through the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.

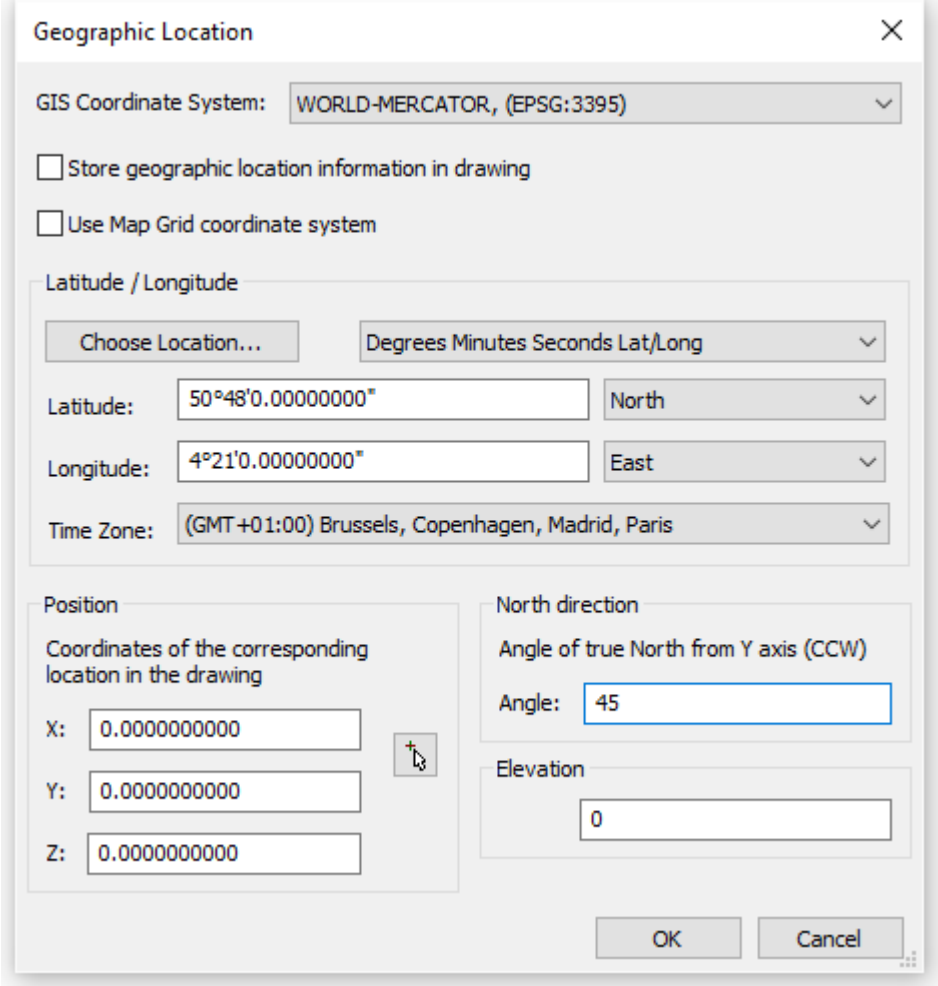
Geographiclocation

Sets the geographic location of the drawing.

Accessing the Command

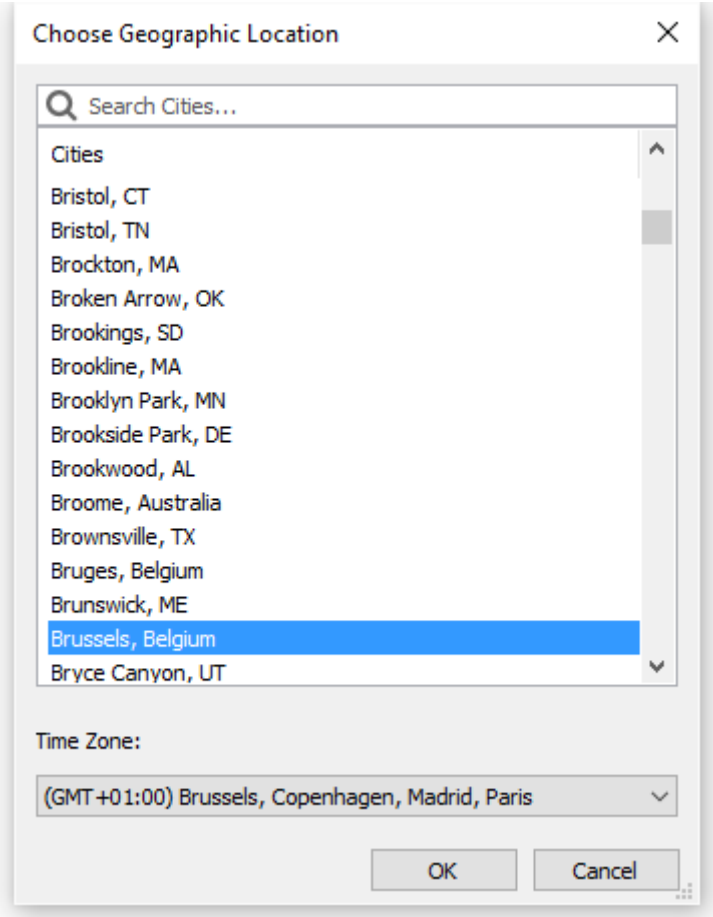
command bar: *geographiclocation*
alias: *geo*
toolbar: *Render* |

: geographiclocation
Displays a dialog box:



Settings

Setting	Description
GIS Coordinate System	Choose a coordinate system from the list.
Store geographic location	If checked, a red dot indicates the position of the selected location in the drawing.

information in the drawing	
Use Map Grid coordinate system	
Choose Location	Displays the *Choose Geographic Location* dialog box.  • Choose a city from the *Cities* drop-down list. Optionally start typing a name in search field. The list will be filtered 'as-you-type'. • Choose a time zone from the Time Zone drop down list. • Click the *OK* button to confirm your choice and close the dialog box.
Latitude	Sets the latitude of the drawing in either Degrees/Min./Sec. or decimal format. The valid range is 0 to 90. Choose *North* or *South* from the latitude option button. The value is saved to the LATITUDE system variable.
Longitude	Sets the longitude of the drawing in either Degrees/Min./Sec. or decimal format. The valid range is 0 to 180. Choose *East* or *West* from the longitude option button. The value is saved to the LONGITUDE system variable.
Timezone	Choose a time zone from the Time Zone drop down list. The setting is saved to the TIMEZONE system variable.

Position	Defines the position of the selected location in the drawing. Either type the coordinates in the X-, Y- and Z-fields or click the *Pick Point* button ( ) and identify a point in the drawing. Optionally, type a value in the *Elevation* field.
North Direction Angle	Specifies the angle of the sun from north, in the context of the world coordinate system. The setting is saved to the NORTHDIRECTION system variable.

Procedures

Managing geographic datasets

Related Commands

Render - calculates a photorealistic image of the 3D model.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
MapConnect - allows to set up a connection with a Web Map Service through a dialog box.

Gradient

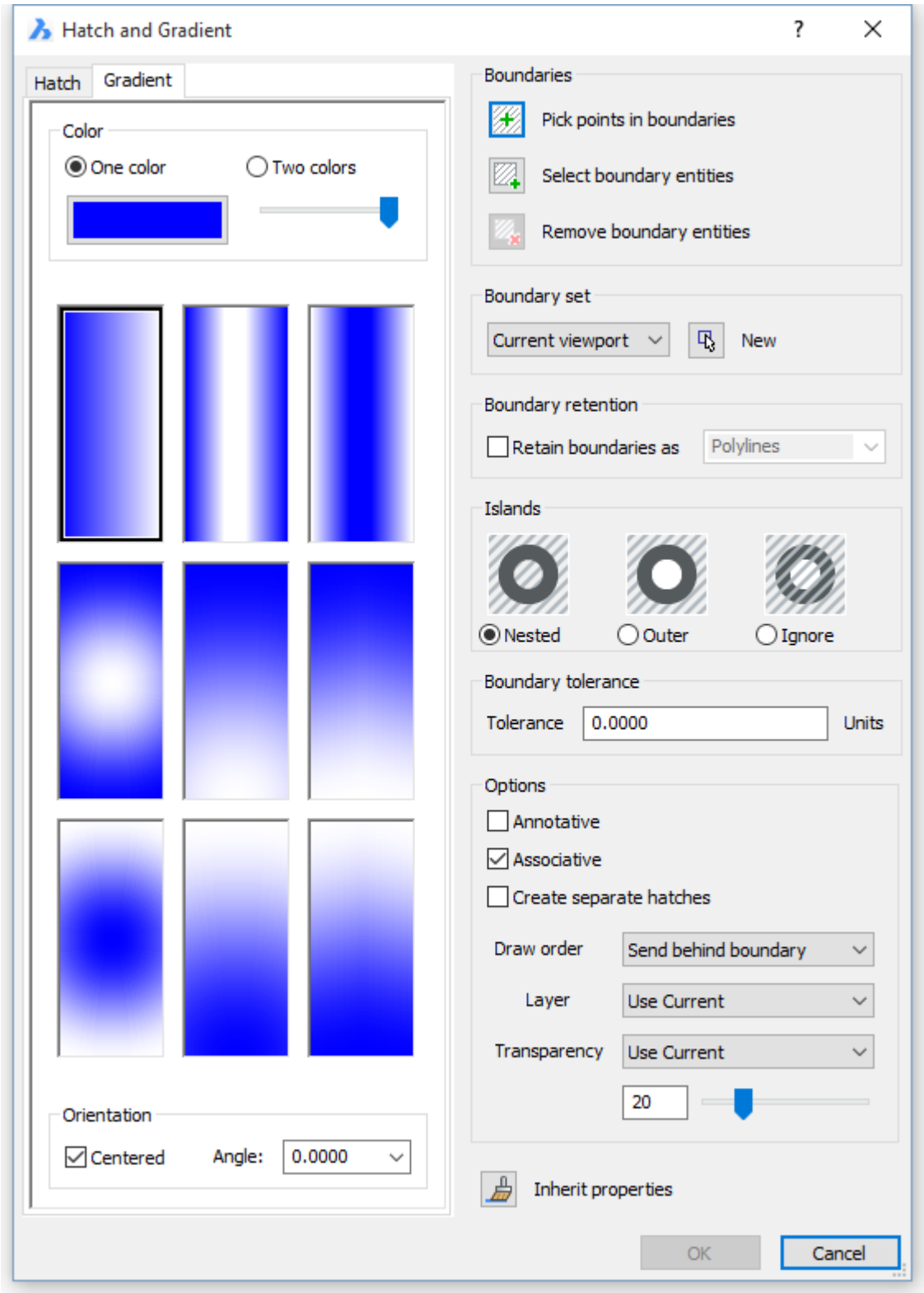
Fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box.

Accessing the Command

command bar: *Gradient*
aliases: none
menu bar: *Draw | Gradient*
toolbar: *Draw |*

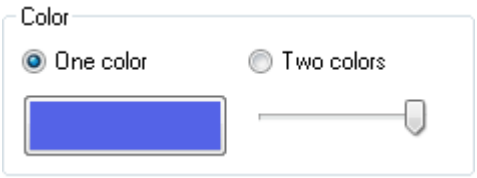
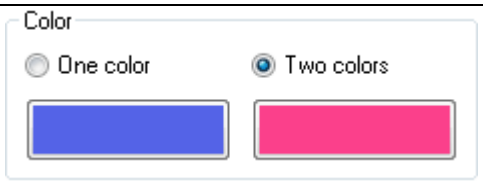
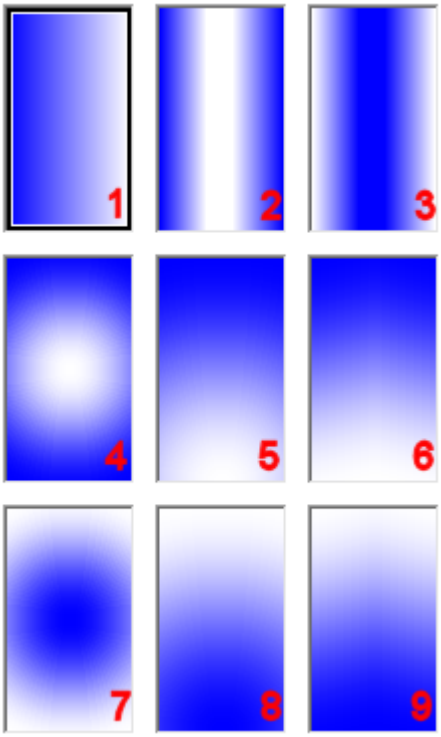
: ***Gradient***

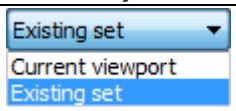
Displays a dialog box:



Choose options, and then click OK.

Command Options

Color Options	Description
	• One color: • Click the colored tile to choose a different base color. • Press and hold the left mouse button to move the slider: • Move the slider to the far right to create a 'color to white' gradient fill. • Move the slider to the far left to create a 'color to black' gradient fill.
	Two colors: Click the colored tiles to choose a different colors.
	Gradient fill pattern name: Click a pattern. The options are: Linear (1) - Cylindrical 2() - Inverted Cylindrical (3) Spherical (4) - Hemispherical (5) - Curved (6) Inverted Spherical (7) - Inverted Hemispherical (8) - Inverted Curved (9)
	Orientation: Centered: The gradient fill is created symmetrical; Not centered: The gradient fill is built up from the left of the boundary. Angle: Specifies the angle of the gradient fill, relative to the current UCS.
Boundaries	
Pick points in boundaries	Specifies the closed areas (boundaries) in which to place the hatch pattern; temporarily dismisses the dialog box, and then prompts you at the command bar:

	Select a point to define a boundary or hatch area - pick a point inside a closed area. Notice that BricsCAD immediately draws the hatch, allowing you to preview its scale factor, angle, origin, and other parameters. The prompt repeats, allowing you to choose additional areas to hatch. When done selecting areas, press Enter: Select a point to define a boundary or hatch area - press Enter to return to the dialog box. You cannot pick an area that already contains a hatch pattern, or areas that are not closed.
Select boundary entities	Selects the entities that make up a hatch pattern boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select entities - use any selection method to choose one or more entities. The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter: Select entities - press Enter to return to the dialog box. This option allows you to hatch closed areas that already contain a hatch; it does not hatch open areas.
Remove boundary entities	Prompts you to remove entities from the detected boundary set.
Boundary Set	
	Specifies where BricsCAD should search for entities that make up the hatch boundary: • Current Viewport - searches all entities in the current viewport. • Existing Set - searches only the current selection set for possible boundaries.
New	Creates a new selection set of entities that make up the pattern boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select entities - use any selection method to choose one or more entities. The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter: Select entities - press Enter to return to the dialog box. The next step is to click the Pick Points in Boundaries button to choose the hatch area. If a selection exists when you launch the *Gradient* command, and if PICKFIRST is *ON*, the *Hatch and Gradient* dialog box immediately reopens when you click the *New* button.

Boundary Retention	
Retain boundaries as	Toggles whether boundaries are retained: • Yes - keeps the boundary in the drawing after this command is finished. • No - removes the boundary after the hatch is created.
Polylines	Specifies the entity used for the boundary: Polylines - makes the boundary from a polyline.
Islands	
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.
Boundary Tolerance	
Tolerance	Specifies the largest gap BricsCAD ignores when hatching a boundary that is not fully closed. The gap tolerance is saved in the HPGAPTOL system variable.
Options	
Annotative	Sets the Annotative property of the hatches.
Associative	Toggles the associativity of hatch patterns: • On - associative; when the boundary is changed, the pattern automatically updates itself. • Off - not associative; when the boundary is changed, the pattern remains as originally placed. • The Associative option is not available for annotative hatches.
Create separate hatches	If checked, creates a separate gradient entity for each boundary in the selection. If not checked, creates a compound gradient entity.
Draw order	Specifies where the pattern should be placed relative to other entities: • None - draw order is ignored. • Send to Back - pattern is behind (underneath) all other entities. • Send to Front - pattern is in front (on top) of all other entities. • Send Behind Boundary - pattern is placed behind the boundary. • Bring in Front of Boundary - pattern is placed in front of the boundary.
Layer	Allows to create the gradient fill on a specified layer, rather than on the current layer. Click the list button, then select a layer.

Transparency	Allows to set the Transparency property of the gradient fill. The options are: • Use current - apply the current transparency value as defined by the CETRANSPARENCY system variable. • By Layer - **apply the value of the transparency property of the layer.** • By Block - transparency value is controlled by block. • Specify value - type a value or use the slider to set a value between 0 (fully opaque) and 90. The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off. The Transparency value for new hatches is saved by the HPTRANSPARENCY system variable.
Inherit properties	Copies the properties of another, existing gradient entity; temporarily dismisses the dialog box, and then prompts you at the command bar: Select hatch entity to copy properties from: *(Use any selection method to choose a single hatch pattern.)* This button is useful for quickly setting options of a new hatch based on existing hatches.

Procedures

Defining the MEASUREMENT setting
Defining the MAXHATCH setting
General procedure to create hatches and gradient fills
To define gradient fill properties
To define the boundary for a hatch or gradient fill
To edit a hatch or gradient fill

Grips Editing

Gradient fills can be edited directly through grips:

1. Select the hatch. Notice that it has one grip.
2. Drag the grip to move the hatch.

Related Commands

Hatch - places hatch patterns through a dialog box.
-BHatch and -Hatch - places hatch patterns through the command bar.
Boundary and -Boundary place a polyline boundary inside closed areas.
HatchEdit - edits hatches and gradient fills in a dialog box.
-HatchEdit - edits the boundaries of hatch and gradient entities; displays prompts in the command bar.
Properties - changes the properties of hatch patterns.
Settings - specifies the default parameters for hatch patterns.
Solid - draws 3- and 4-side solid-filled areas.

GradientBkgOff

Switches the gradient background color of the current viewport off. Restores the default view background.

Accessing the Command

command bar: *gradientbkgoff*

: ***gradientbkgoff***
Displays no prompts you in the command bar.

Command Options

No Options

Related Commands

Background - sets the model space background for the current viewport and saved views.
GradientBkgOn - restores the default gradient background of the current viewport.
View - creates and sets named views through the Drawing Explorer.
-View - creates and sets named views through the command bar.

GradientBkgOn

Restores the default gradient background of the current viewport.

Accessing the Command

command bar: *gradientbkgon*

: ***gradientbkgon***
Displays no prompts you in the command bar:

Command Options

No options.

Related Commands

Background - sets the model space background for the current viewport and saved views.
GradientBkgOff - switches the gradient background color of the current viewport off. Restores the default view background.
View - creates and sets named views through the Drawing Explorer.
-View - creates and sets named views through the command bar.

GraphScr

Switches from the text windows to the graphics windows (short for "graphics screen").

Accessing the Command

command bar: *graphscr*
transparent: *'graphscr*
shortcut: *F2*

: ***graphscr***
No prompt in the command bar; brings the drawing window (graphics screen) to the foreground.

Command Options

No options.

Related Commands

TextScr - brings the text window to the foreground.

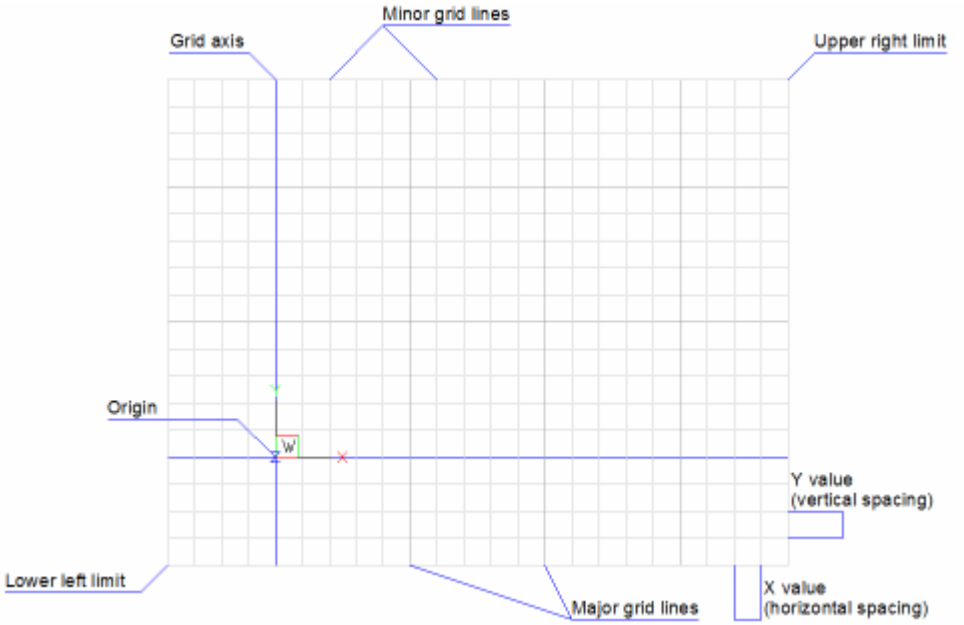
Grid

Toggles the display of the grid; sets the grid properties.

Accessing the Command

command bar: *grid*
transparent: *'grid*
shortcut: *F7*
alias: *g*
menu bar: *Settings | Reference Grid*
toolbar: *Settings |* ▦
status bar: *GRID*

: ***grid***
Prompts you in the command bar:
Grid is off. Enter grid spacing or [grid ON/sync to Snap/Aspect] <10>: ***(Enter an option.)***



The extents of the grid are controlled by the Limits command if the *Display beyond limits area* flag of the GRIDDISPLAY system variable is not set.

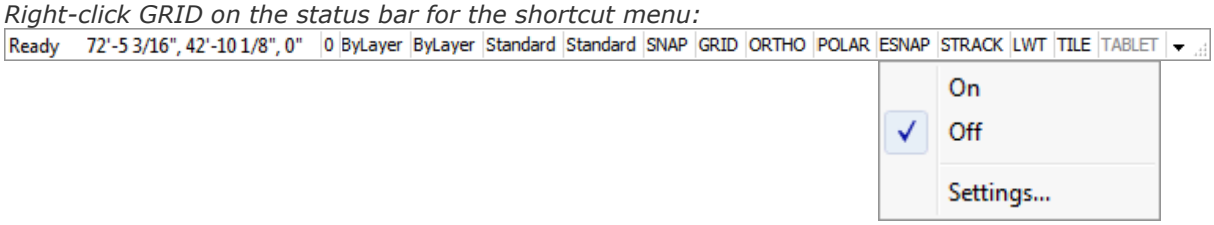
Command Options

Option	Description
grid ON grid OFf	Allows to toggle the grid on/off.
Aspect	Changes the aspect ratio of the grid; prompts you: Horizontal Spacing - specify the x distance between grid lines or dots. Vertical Spacing - specify the y distance between grid lines or dots. To change the grid to isometric mode, use the Snap command.
Enter grid spacing	Makes the x and y grid spacing the same; enter a positive number.

Sync to snap	Synchronizes the grid spacing with the current snap spacing. Do one of the following: • Type *snap* or *s* at the command prompt, then press Enter. • Press Enter. The grid spacing stays synced with the current snap spacing until the Grid Spacing or Aspect option is used to set grid spacing different from snap spacing.

Status Bar Menu

Right-click GRID on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on grid display.
Off	Turns off grid display.
Settings	Opens the Snap/Grid section of the Settings dialog box. **Snap/Grid** Grid mode: Grid on Grid unit: 0.5000, 0.5000 X: 0.5000 Y: 0.5000 Grid major: 5 Grid display: 0x0001 (1) 0x0001: Display beyond LIMITS area 0x0002: Adaptive grid display 0x0004: Allow subdivision below grid spacing 0x0008: Follow Dynamic UCS Grid style: 0x0000 (0) 0x0001: Dotted grid in 2D model space 0x0002: Dotted grid in Block Editor 0x0004: Dotted grid in Sheet/Layout Grid axis color: Blue Grid major color: 253 Grid minor color: 254 Grid XYZ tint: 0x0000 (0) 0x0001: Apply XYZ tint to grid axis lines 0x0002: Apply XYZ tint to major grid lines 0x0004: Apply XYZ tint to minor grid lines Snap angle: 0
Grid mode	Toggles grid display: • On - turns on grid display. • Off - turns off grid display.

Grid unit	Specifies the distance between grid lines or dots: • X - spacing between dots in the x direction. • Y - spacing between dots in the y direction. There is no grid in the z direction.
Grid Major	Specifies the frequency of major versus minor grid lines.
Grid Display	Specifies how the grid displays. The options are: • Display beyond limits area: displays the grid all over the graphic screen. If this option is off, the display of the grid is limited to the area defined by the LIMMIN and LIMMAX system variables, set by the Limits command. • Adaptive grid display: displays less grid lines if the grid spacing is too small to display. If this option is off a message displays in the command window when it is not possible to display the grid properly. • Allow subdivision below grid spacing: displays more grid lines when you zoom in. If this option is off the distance between the minor grid lines equals the current grid spacing. • Follow dynamic UCS (not implemented yet): the grid displays in the XY-plane of the dynamic UCS . If this option is off, the grid displays in the XY-plane of the current UCS.
Grid Style	Specifies whether the grid displays with dots or lines. The options are: • Dotted grid in 2D model space: Dots display instead of lines if the view mode is set to 2D Wireframe. In all other view modes the grid always displays with lines (see the Shademode command). • Dotted grid in Block Editor (not implemented yet). • Dotted grid in Sheet/Layout: Dots display instead of lines in paper space layouts.
Grid axis color	Specifies the color of the grid axis lines and the major and minor grid lines.
Grid XYZ tint	Specifies whether the XYZ tints are applied over the grid line colors. XYZ tints are defined by the COLORX, COLORY and COLORZ system variables.
Snap angle	Defines the rotation of snap, grid and crosshairs for the current viewport, relative to the current UCS. If the snap angle is different from zero only dotted grids display; grid lines do not display.

Related Commands

Snap - sets the snap spacing.
Limits - limits the extent of the grid dots.
Settings - sets grid parameters through a dialog box.

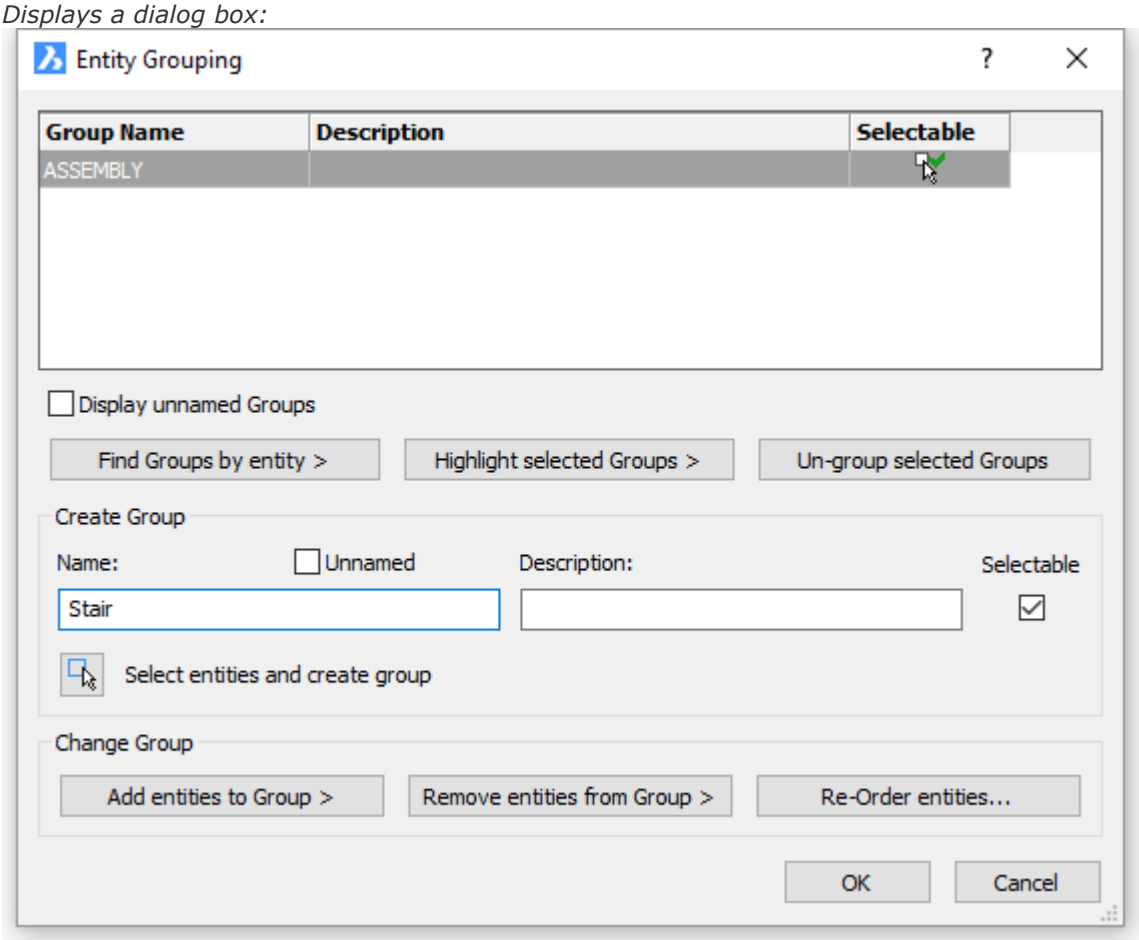
Group

Creates and modifies named groups of entities.

Accessing the Command

command bar: *group*
menu bar: *Tools | Group*
toolbar: *Tools |*

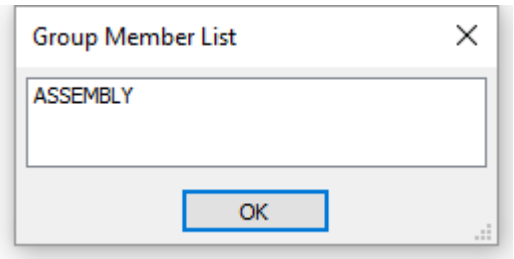
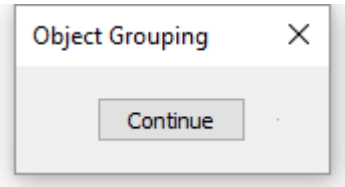
: ***group***
Displays a dialog box:



*Create or edit a named group, and then click **OK**.*

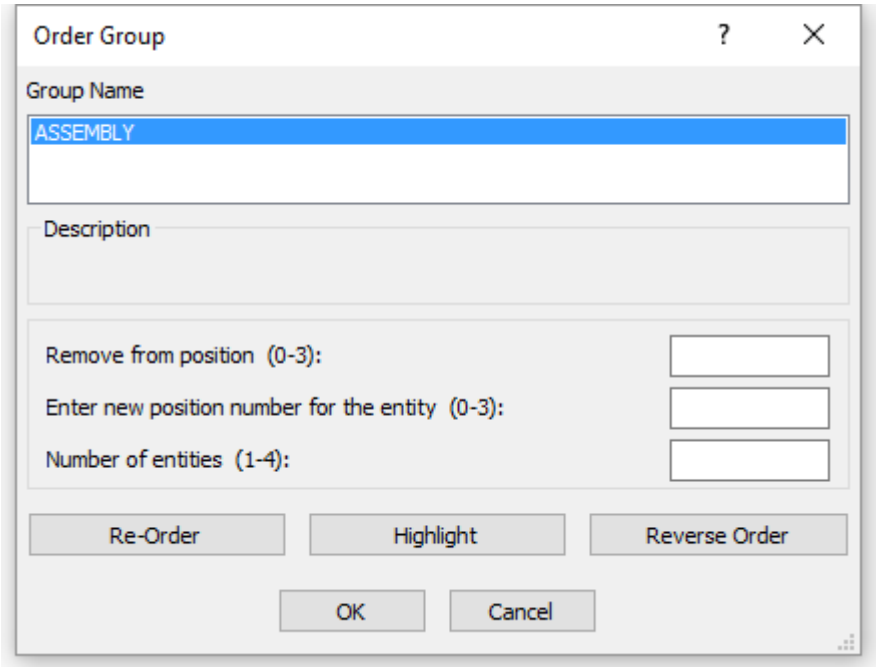
Command Options

Option	Description
Display Unnamed Groups	Toggles the display of unnamed groups: • Yes - displays unnamed groups created by BricsCAD. • No - displays only named groups created by users.
Find Groups by Entity	Reports the name(s) of groups when you select entities in the drawing. Temporarily removes the dialog box, and then prompts: Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to continue.

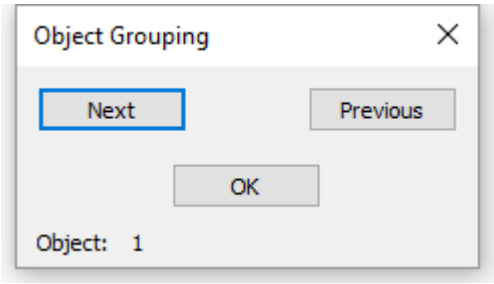
	Displays the Group Member List dialog box:  Click OK to close this dialog box and to return to the Entity Grouping dialog box.
Highlight Selected Groups	Highlights entities that belong to the selected group. 1. In the Entity Grouping dialog box, choose one or more groups in the Group Name list. 2. Click the Highlight Selected Groups button. 3. Notice that the dialog box disappears and that entities are highlighted.  4. Click OK to return to the Entity Grouping dialog box.
Un-group Selected Groups	Removes group status from selected entities. Select groups in the Group Names list, and then click this button. The names disappear from the list.
Create Group	
Name	Specifies the names of groups.
Description	Specifies an optional description of the group.
Unnamed	Toggles group naming: • Yes - allows unnamed groups to be created; the Name field is grayed out. • No - requires that groups be named. Unnamed groups are assigned the prefix *A following by an incremented number, such as *A2.
Selectable	Toggles the selectability of a group if the *Group Selection* option of the PICKSTYLE system variable is set (PICKSTYLE = 1 or 3. • Yes - allows only the group to be selected; entities that are part of a group cannot be selected. • No- allows individual members of a group to be selected.
Select Entities	Dismisses the dialog box temporarily, and then prompts you at the command bar: Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to return to the dialog box.

	Back in the dialog box, the selected entities become part of the group.

Change Group	
Add Entities to Group	Adds entities to the selected group; prompts you at the command bar: Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to return to the dialog box.
Remove Entities from Group	Removes entities from the selected group; prompts you at the command bar: Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to return to the dialog box.
Re-Order Entities	Changes the order of entities in groups; displays the Order Group dialog box.



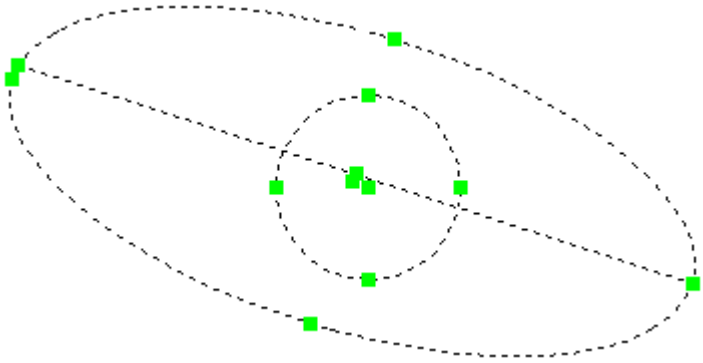
Order Group	**Description**
Group Name	Lists the names of named and unnamed groups.
Remove From Position	Specifies the position number of the entity, which is to be renumbered.
Enter New Position Number for the Entity	Specifies the new position number for the entity.
Number of Entities	Specifies the range of entities to reorder.
Re-Order	Applies the re-ordering dictated by the previous three fields.
Highlight	Highlights entities in the group; dismisses the Order Group dialog box, and then displays the following one:

	 **Next** - highlights the next entity of the group. **Previous** - highlights the previous entity. **OK** - returns to the Order Group dialog box.
Reverse Order	Reverses the order of entities in the group.

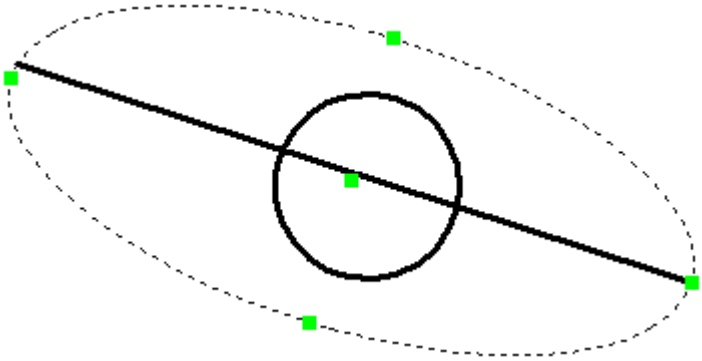
Grips Editing

Groups can be edited directly through grips in two ways:

Selectable - select one entity in the group, and all entities are selected (show highlights and grips).



Not Selectable - select one entity and only that entity is selected, such as the ellipse shown below.



Procedures

To create a group
To modify a group
To ungroup entities
To change the order of entities

Related Commands

-Group - creates and modifies named groups of entities, at the command line.
Block - groups entities into a block (symbol), through a dialog box
-Block - groups entities into a block (symbol), at the command line.

-Group

Creates and modifies named groups of entities, at the command line.

Accessing the Command

command bar: *-group*

: ***-group***
Prompts you in the command bar:
Enter group option [? to list/Order/Add/Remove/Un-group/REName/Selectable/Create]
<Create>: *(Enter an option.)*

Command Options

Option	Description
? to list	Lists named and unnamed groups in the drawing, and then exits the command. Sample listing: Group name: SIDEVIEW Group description: Side View of Boiler Selectable: Yes Unnamed groups are prefixed with *A, followed by an incremented number, such as *A3.
Order	Reverses the order of entities in a group; prompts you: Enter a group name or ? to list - specify the name of a group. Reverse order - enter Y or N.
Add	Adds entities to a group; prompts you: Enter a group name or ? to list - specify the name of a group. Select entities - choose one or more entities, and then press Enter.
Remove	Removes entities from a group; prompts you: Enter a group name or ? to list - specify the name of a group. Select entities - choose one or more entities, and then press Enter.
Un-group	Removes entities from the group, and then removes the group from the drawing; prompts you: Enter a group name or ? to list - specify the name of a group.
REName	Renames groups; prompts you: Enter a group name or ? to list - specify the name of a group. Enter new name - specify a different name for the group.
Selectable	Toggles the selectability of groups; prompts you: Enter a group name or ? to list - specify the name of a group. This group is selectable, do you want to change it -- enter Y or N.
Create	Creates new groups; prompts you: Enter a group name or ? to list [* to create unnamed] - enter a name, or press * for an unnamed group. Enter description - enter a description, or press Enter for no description. Select entities - choose one or more entities. Select entities - choose more entities, or press Enter to create the group.

Related Commands

Group - creates and modifies named groups of entities, through a dialog box
Block - groups entities into a block (symbol), through a dialog box
-Block - groups entities into a block (symbol), at the command line.

H

Hatch

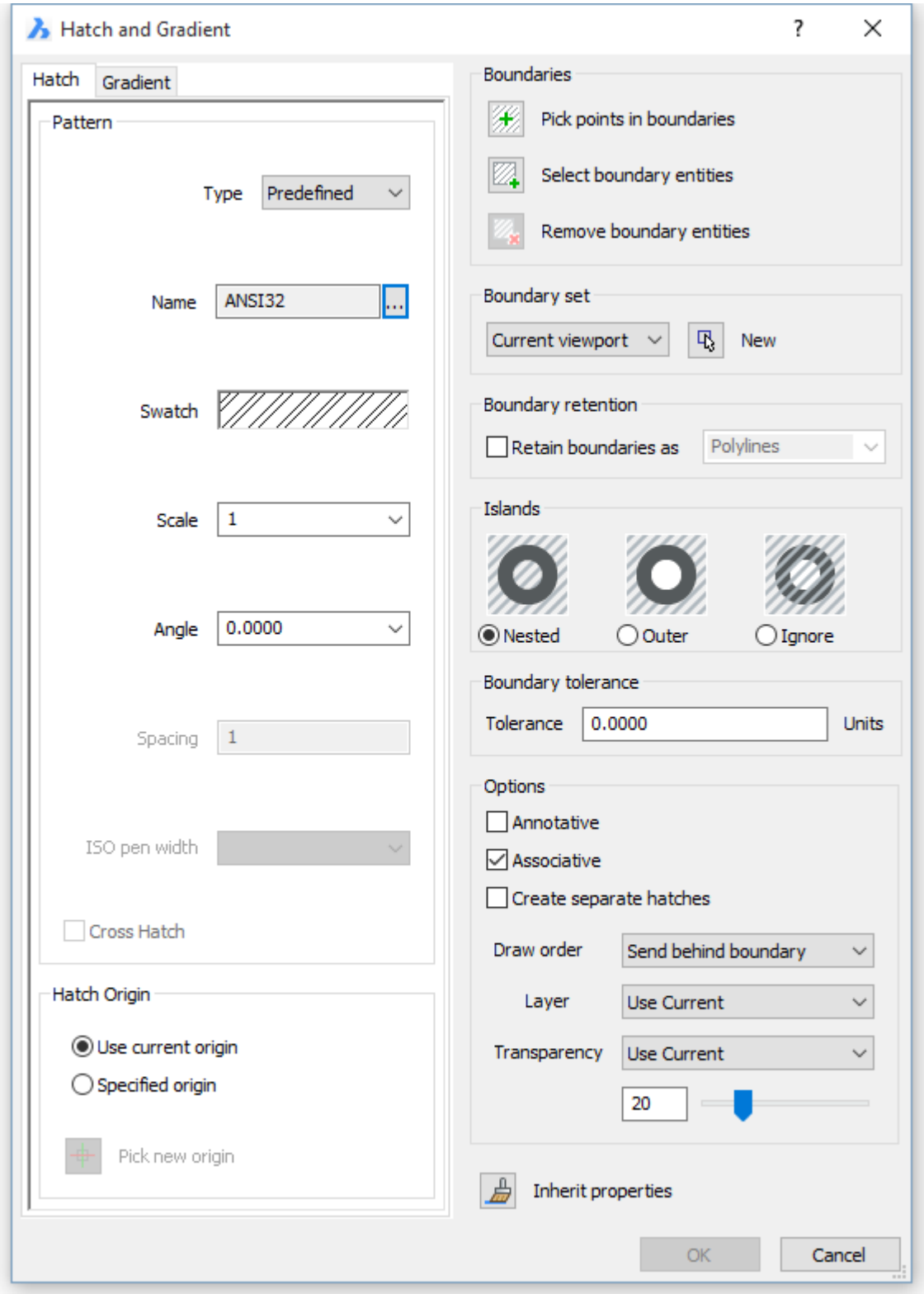
Fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box.

Accessing the Command

command bar: *hatch*
aliases: *ba, h, bhatch*
menu bar: *Draw | Hatch*
toolbar: *Draw |*

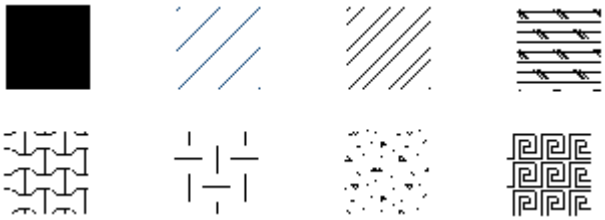
: ***hatch***

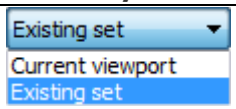
Displays a dialog box:



Choose options, and then click OK.

Command Options

Pattern Options	Description
Type	Selects the type of hatch pattern: • User Defined - uses the pattern defined in this dialog box with the Angle, Spacing, and Cross Hatch options. • Predefined - uses patterns defined in *iso.pat* (metric units - MEASUREMENT = ON) and *default.pat* (imperial units - MEASUREMENT = OFF). • Custom - uses patterns defined in *.pat files (one pattern definition per file). BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (Support File Search Path) system variable.
Name	Specifies the name of the predefined hatch pattern; type a name, or else click the ... button to display the Hatch Pattern Palette dialog box.
Swatch	Click to display the Hatch Pattern Palette dialog box; choose a pattern. Examples of patterns: 
Scale	Specifies the scale factor of the pattern; choose a preset scale factor from the droplist, or else enter a custom factor. The scale factor for hatch patterns is generally the same as that for text and linetypes.
Angle	Specifies the angle of the pattern; choose a preset angle from the droplist, or else enter a custom angle. This angle is measured relative to the positive x-axis of the current UCS.
Spacing	(For user defined patterns only.) Specifies the spacing between lines.
ISO pen width	(For ISO hatch patterns only.) Specifies the lineweight.
Cross hatch	(For user defined patterns only.) Determines whether the pattern is crossed (repeated at 90 degrees to the original).
Hatch Origin	
Use current origin	Uses the drawing's origin (0,0) as the origin of the hatch pattern.
Specified origin	Uses another point for the pattern's origin, chosen through the Pick New Origin button. Use this option to begin the hatch pattern at the corner of a wall or other location where the placement of the pattern is critical.
Pick new origin	(For Specified origin only.) Dismisses the dialog box temporarily, and then prompts you at the command bar: Specify origin point - pick a point in the drawing.
Boundaries	
Pick points in boundaries	Specifies the closed areas (boundaries) in which to place the hatch pattern; temporarily dismisses the dialog box, and then prompts you at the command bar: Select a point to define a boundary or hatch area - pick a point inside a closed area.

	Notice that BricsCAD immediately draws the hatch, allowing you to preview its scale factor, angle, origin, and other parameters. The prompt repeats, allowing you to choose additional areas to hatch. When done selecting areas, press Enter: Select a point to define a boundary or hatch area - press Enter to return to the dialog box. You cannot pick an area that already contains a hatch pattern, or areas that are not closed.
Select boundary entities	Selects the entities that make up a hatch pattern boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select entities - use any selection method to choose one or more entities. The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter: Select entities - press Enter to return to the dialog box. This option allows you to hatch closed areas that already contain a hatch; it does not hatch open areas.
Remove boundary entities	Prompts you to remove entities from the detected boundary set.
Boundary Set	
	Specifies where BricsCAD should search for entities that make up the hatch boundary: • Current Viewport - searches all entities in the current viewport. • Existing Set - searches only the current selection set for possible boundaries.
New	Creates a new selection set of entities that make up the pattern boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: Select entities - use any selection method to choose one or more entities. The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter: Select entities - press Enter to return to the dialog box. The next step is to click the Pick Points in Boundaries button to choose the hatch area. If a selection exists when you launch the *Hatch* command, and if PICKFIRST is *ON*, the *Hatch and Gradient* dialog box immediately reopens when you click the *New* button.
Boundary Retention	
Retain boundaries as	Toggles whether boundaries are retained: • Yes - keeps the boundary in the drawing after this command is finished. • No - removes the boundary after the hatch is created.
Polylines	Specifies the entity used for the boundary: Polylines - makes the boundary from a polyline.
Islands	
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.

Boundary Tolerance	
Tolerance	Specifies the largest gap BricsCAD ignores when hatching a boundary that is not fully closed. The gap tolerance is saved in the HPGAPTOL system variable.
Options	
Annotative	Sets the Annotative property of the hatches. Applies the current annotative scale (see the CANNOSCALE system variable).
Associative	Toggles the associativity of hatch patterns: • On - associative; when the boundary is changed, the pattern automatically updates itself. • Off - not associative; when the boundary is changed, the pattern remains as originally placed. • The Associative option is not available for annotative hatches.
Create separate hatches	If checked, creates a separate hatch entity for each boundary in the selection. If not checked, creates a compound hatch entity.
Draw order	Specifies where the pattern should be placed relative to other entities: • None - draw order is ignored. • Send to Back - pattern is behind (underneath) all other entities. • Send to Front - pattern is in front (on top) of all other entities. • Send Behind Boundary - pattern is placed behind the boundary. • Bring in Front of Boundary - pattern is placed in front of the boundary.
Layer	Allows to create the hatch on a specified layer, rather than on the current layer. Click the list button, then select a layer.
Transparency	Allows to set the Transparency property of the hatch. The options are: • Use current - apply the current transparency value as defined by the CETRANSPARENCY system variable. • By Layer - **apply the value of the transparency property of the layer.** • By Block - the transparency value is controlled by block. • Specify value - type a value or use the slider to set a value between 0 (fully opaque) and 90. The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off. The Transparency value for new hatches is saved by the HPTRANSPARENCY system variable.
Inherit properties	Copies the properties of another, existing hatch pattern; temporarily dismisses the dialog box, and then prompts you at the command bar: Select hatch entity to copy properties from: (Use any selection method to choose a single hatch pattern.)

	The prompt repeats, allowing you to choose a different hatch entity. When done, press Enter: Select hatch entity to copy properties from: (Press Enter to return to the dialog box.) This button is useful for quickly setting options of a new hatch based on existing hatches.

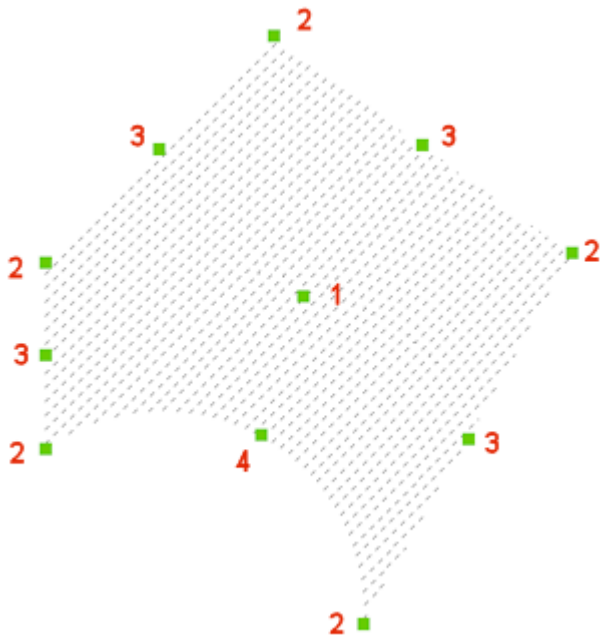
Procedures

Defining the MEASUREMENT setting
Defining the MAXHATCH setting
General procedure to create hatches and gradient fills
To define hatch properties
To define the boundary for a hatch or gradient fill
To edit a hatch or gradient fill

Grips Editing

Non-associative hatches can be edited directly through grips:

1. Select the hatch.
   Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.



1. Center grip: moves the hatch
2. Vertex grip: moves the vertex
3. Midpoint grip: moves the boundary segment
4. Arc midpoint grip: modifies the arc radius

Related Commands

BHatch - places hatch patterns through a dialog box.
Boundary and -Boundary place a polyline boundary inside closed areas.
Gradient - places a gradient fill through a dialog box.
-Hatch - places hatch patterns through the command bar.
HatchEdit - edits hatches and gradient fills in a dialog box.
-HatchEdit - edits the boundaries of hatch and gradient entities; displays prompts in the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Properties - changes the properties of hatch patterns.
Settings - specifies the default parameters for hatch patterns.
Solid - draws 3- and 4-side solid-filled areas.

-Hatch

Fills closed areas with repeating patterns; displays prompts at the command bar.

Accessing the Command

command bar: *-hatch*
aliases: *-bh, h, -bhatch*
Quad: *2D Editing* |

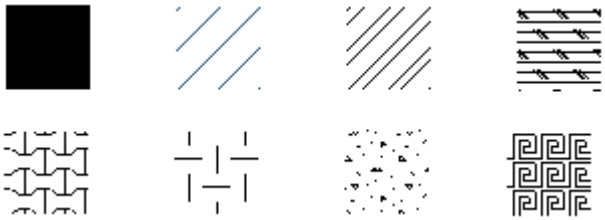
: ***-hatch***
Prompts you in the command bar:

Current hatch pattern: ANSI31

Specify internal point or
[Properties/Select/Remove/Advanced/Draworder/Origin/ANnotative/Layer/Transparency/Undo]:
(Pick a point, or enter an option.)
Selecting everything visible...*

Specify internal point or
[Properties/Select/Remove/Advanced/Draworder/Origin/ANnotative/Layer/Transparency/Undo]:
(Pick another point, or enter an option, or press Enter to create the hatch.)

*** Entities which are not visible in the current viewport when the command is launched cannot be selected. Partly visible entities can be selected though.**



Examples of hatch patterns.

Command Options

Option	Description
Current hatch pattern: ANSI31	Reports the name of the current hatch pattern; you can change the pattern name with the Properties option.
Specify internal point	Pick a point inside a closed area; BricsCAD creates an invisible boundary, and then hatches the area. You can pick more than one area to hatch at a time.
Properties	
Enter a pattern name	Specifies the name of the hatch pattern.
? to list patterns	Lists the names of all patterns; reminds you of their names. Press F2 to see the list in the prompt history window.
Solid	Specifies a solid filled hatch.
User defined	Allows you to define simple hatch patterns based on angle, scale, and crosshatching.
Proceed	Skips the next two prompts, and returns to the original prompt.
Scale for pattern	Specifies the scale factor of the hatch pattern. The scale factor is usually the same as that of text and linetypes.
Angle for pattern	Specifies the angle at which the pattern is drawn.

User Defined	
Proceed	Skips the following prompts, and returns to the original prompt.
Space between standard pattern lines	Specifies the distance between hatch lines.
Angle for lines	Specifies the angle at which the pattern is drawn.
Cross-hatch area?	Applies the pattern a second time at 90 degrees to the first.
Select	Prompts you to select the entities making up the boundary: Select entities - select one or more entities. Select entities - press Enter to return to original prompt.
Remove	Allows to remove boundary entities. Prompts you to select island (interior) entities, which will be hatched through.
Advanced	Prompts you: Enter an option [Boundary set/Retain boundary/Island detection/Specify style/Set associativity(A)/Boundary tolerance(T)/Seperate hatches(H)]:
Boundary set	Specifies the entities to consider for creating boundaries: Specify candidate set for boundary [New selection/All objects(E)]<Everything>: - enter Y or N. • Everything - selects all entities in the current viewport. • New - prompts you to select entities.
Retain boundary	Determines whether the temporary boundary created by BricsCAD is retained after the command ends: Retain derived boundaries? [Yes-Retain boundariess(Y)/No-Discard boundary(N)]<No>: - enter Y or N. • Yes - boundary is kept in the drawing. • No - boundary is erased.
Island detection	Toggles whether islands are hatched over or not: Do you want island detection? [Yes-Detect islands(Y)/No-Set ray casting(N)]<Yes>: - enter Y or N. • Yes - islands (interior boundaries) are not hatched. • No - islands are hatched over.
Style	Specifies how islands are treated: Enter hatching style: [Nested/Outer/Ignore]<Nested>: - enter N, O, or I. • 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.
Associativity	Toggles whether hatches are associative: Associativity hatch? [Yes-Associative hatch(Y)/No-Nonassociative hatch(N)]<Yes>: - enter Y or N. • Yes - hatches are associative; when their boundary changes, they automatically update.

	• No - hatches are not associative; when the boundary changes, the hatch keeps its original shape.
Tolerance	Specifies the largest gap that BricsCAD will close when hatching open areas: Specify a boundary gap tolerance value <0>: - enter a value.

Separate Hatches	Create separate hatches? [Yes/No-Do not separate(No)]:<No>: - enter Y or N. • Yes - creates seperate hatches if the hatch is composed of multiple closed boundaries • No - creates a single hatch entity.
Draw order	Specifies whether the hatch pattern appears visually above or below overlapping entities. Prompts you: Enter an option [None/Send to back(B)/Send to front(F)/Send behind boundary(H)/Bring in front of boundary(D)]: - enter an option. • **None** - ignores draw order. • **send to Back** - places the hatch pattern behind (below) other entities; this prevents the hatch from obscuring other entities. • **bring to Front** - places the pattern in front (on top) of other entities. • **send beHind boundary** - places the pattern behind its boundary; this makes it easier to select the boundary. • bring in front of bounDary - places the pattern in front of its boundary.
Origin	Prompts for a new hatch origin point: New value for the hatch origin <0.0000,0.0000>: - enter x,y coordinates, or pick a point in the drawing. Use entity snaps to pick the origin accurately.
Annotative	Sets the Annotative property of the hatches. Applies the current annotative scale (see the CANNOSCALE system variable). Prompts you: Create annotative hatch [Yes/No] <No> - type Y or N. • Yes - creates annotative hatches. • No - creates non-annotative hatches
Layer	• Allows to create the hatch on a specified layer, rather than on the current layer. • Prompts you: • **Specify layer or . for use current <.>:** • Type the name of the layer you want to create the hatch entity on or press Enter to create the hatch on the current layer.

Transparency	• Allows to set the Transparency property of the hatch. • Prompts you: • Specify transparency [0/90/ByLayer(L)/ByBlock(B)/. for use current(.)] <Use current>: - enter an option. • Use current - apply the current transparency value as defined by the CETRANSPARENCY system variable. • Specify transparency [0,90] - type a value or use the slider to set a value between 0 (fully opaque) and 90. The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off. • By Layer - apply the value of the transparency property of the layer. • By Block - transparency value is controlled by block. The Transparency value for new hatches is saved by the HPTRANSPARENCY system variable.
Undo	Allows to remove already selected boundaries from the selection.

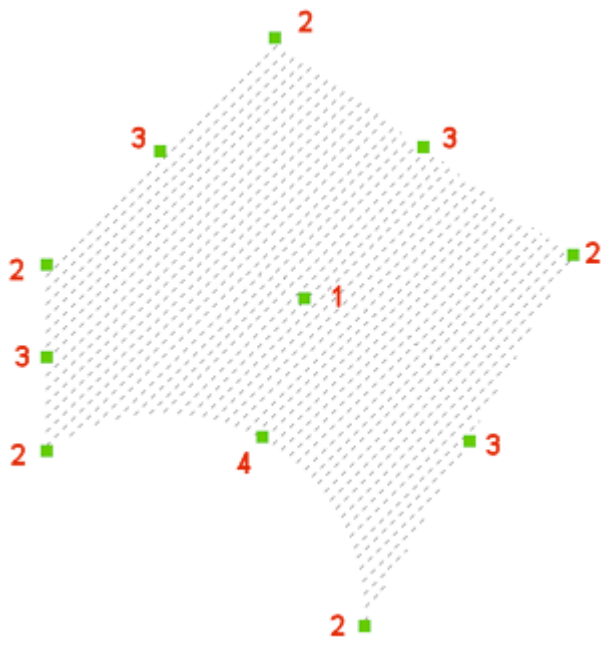
Procedures

Defining the MEASUREMENT setting
Defining the MAXHATCH setting
General procedure to create hatches and gradient fills
To define hatch properties
To define the boundary for a hatch or gradient fill
To edit a hatch or gradient fill

Grips Editing

Hatches can be edited directly through grips:

1. Select the hatch.
   Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.



1. Center grip: moves the hatch
2. Vertex grip: moves the vertex
3. Midpoint grip: moves the boundary segment
4. Arc midpoint grip: modifies the arc radius

Related Commands

Hatch - place hatch patterns through a dialog box.
Boundary and -Boundary place a polyline boundary inside closed areas.
HatchEdit - edits hatches and gradient fills in a dialog box.
-HatchEdit - edits the boundaries of hatch and gradient entities; displays prompts in the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Properties - changes the properties of hatch patterns.
Settings - specifies the default parameters for hatch patterns.
Solid - draws 3- and 4-side solid-filled areas.

HatchEdit

Edits hatches and gradient fills in a dialog box.

Accessing the Command

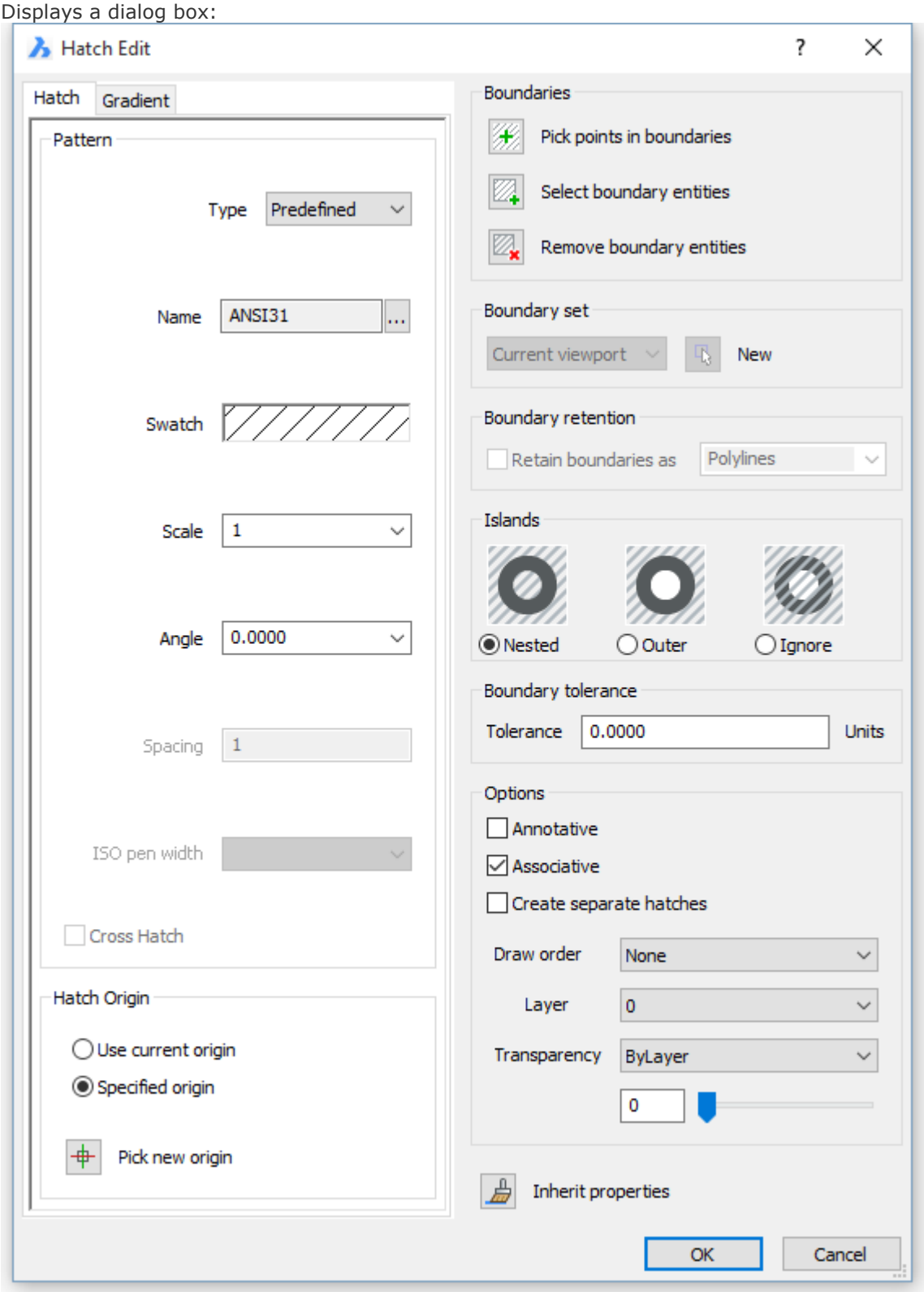
command bar: *hatchedit*
menu bar: *Modify | Hatch edit*
toolbar: *Modify |*

: ***hatchedit***

Prompts you in the command bar:

Select hatch entity: ***(Select a hatch or gradient fill in the drawing)***

Displays a dialog box:



If a hatch is selected, see the Hatch command to edit the selected entity as a linear or a pattern hatch.
If a gradient fill is selected, see the Gradient command to edit the selected entity as a gradient hatch.
Click the *OK* button to apply the changes.

Procedures

To edit a hatch or gradient fill in the properties bar
To edit a hatch in a dialog box
To edit a gradient fill in a dialog box

Related Commands

Gradient - Fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box.
Hatch - fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box.
-Hatch - fills closed areas with repeating patterns; displays prompts at the command bar.
-HatchEdit - Edits the boundaries of hatch and gradient entities; displays prompts in the command bar.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.

-HatchEdit

Edits the boundaries of hatch and gradient entities; displays prompts in the command bar.

Accessing the Command

command bar: *-hatchedit*
Quad cursor menu: *2D Editing* | | |

: ***-hatchedit***
Prompts you in the command bar:
Select hatch entity: *(Select a hatch or a gradient entity in the drawing)*
Enter hatch option [Disassociate/ADd/Remove/ASsociate]: *(Enter an option, then press Enter)*

Command Options

Option	Quad Icon	Description
Disassociate		Removes the *Associative* property of the selected hatch or gradient entity. BricsCAD reports in the command bar: Hatch boundary associativity removed
Add		Adds boundary entities to the selected hatch or gradient entity. Prompts you in the command bar: Specify internal point or [Select entities] **- do one of the following:** • Click inside a closed area. Selecting everything visible... Specify internal point or [Select entities] - Click inside another closed area or press Enter to stop. • Choose *Select entities*. Prompts you: Select entities - Select a closed entity (polyline, circle, ellipse, ...), Entities in set: 1 Select entities - Select another closed entity or press Enter to stop.
Remove		Removes boundary entities from the selected hatch or gradient entity. Boundaries of the selected hatch or gradient entity highlight. Prompts you in the command bar: Select entities - click a boundary Select entities - click another entity or press Enter to stop.

Associate		Associates the selected hatch or gradient entity to a different boundary set. Prompts you in the command bar: Specify internal point or [Select entities] **- do one of the following:** • Click inside a closed area. Selecting everything visible... Specify internal point or [Select entities] - Click inside another closed area or press Enter to stop. • Choose *Select entities*. Prompts you: Select entities - Select a closed entity (polyline, circle, ellipse, ...), Entities in set: 1 Select entities - Select another closed entity or press Enter to stop.

Related Commands

HatchEdit - edits hatches and gradient fills in a dialog box.
-Hatch - fills closed areas with repeating patterns; displays prompts at the command bar.
Hatch - fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box.
HatchGenerateBoundary - generates a boundary around a hatch or gradient fill.
Gradient - fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box.

HatchGenerateBoundary

Generates a boundary around a hatch or gradient fill.

Accessing the Command

command bar: *hatchgenerateboundary*
menu: *Draw | Boundaries | Hatch Generate Boundary*
Quad: *2D Editing |*

: ***hatchgenerateboundary***
Prompts you in the command bar:

Select hatches to generate boundary: (Select a hatch or gradient fill.)
Entities in set: 1

Select hatches to generate boundary: (Select another hatch or gradient fill or press Enter to stop adding entities.)

Generated boundaries for # hatch(es)

A boundary polyline is created around the selected hatches or gradient fills.

Command Options

No options.

Related Commands

BHatch - places hatch patterns through a dialog box.
Boundary and -Boundary place a polyline boundary inside closed areas.
GenerateBoundary - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated.
Gradient - places a gradient fill through a dialog box.
-Hatch - places hatch patterns through the command bar.
HatchEdit - edits hatches and gradient fills in a dialog box.
-HatchEdit - edits the boundaries of hatch and gradient entities; displays prompts in the command bar.

HatchToBack

Sets the draw order of all hatch entities in the drawing to display behind all other entities.

Accessing the Command

command bar: *hatchtoback*
menu bar: *Tools | Draw Order | Send Hatches to Back*
toolbar: *Draw Order |*

: ***hatchtoback***
hatch entity(ies) sent to back

Command Options

No options.

Related Commands

DrawOrder - changes the display order of overlapping entities.
DrawOrderByLayer - sets draw order of overlapping objects through layer names.
TextToFront - sets the draw order of all texts and dimensions in the drawing to display in front of all other entities.

Helix

Creates a 2D spiral or a 3D helix.

Accessing the Command

command bar: *helix*
menu bar: *Draw| Helix*
toolbar: *Polylines |*

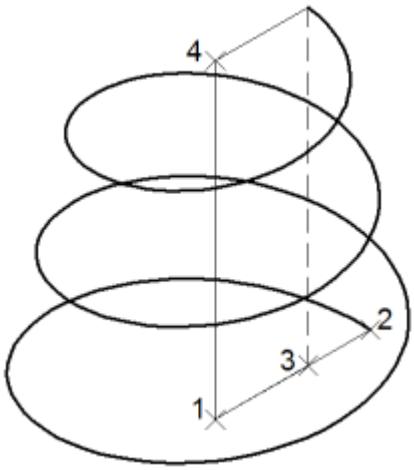
: ***helix***
Prompts you in the command bar:
Number of turns = 3.0 Twist = CCW

Specify center point of base: (Specify a point in the drawing.) (1)

Specify base radius or [Diameter] <default>: (Specify the radius of the base or press Enter to accept the default.) (2)

Specify top radius or [Diameter] <default>: (Specify the radius of the top or press Enter to the top radius equal to the base radius.) (3)

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <default>: (Specify the height of the helix or enter an option or enter 0 to create a 2D spiral) (4)



Command Options

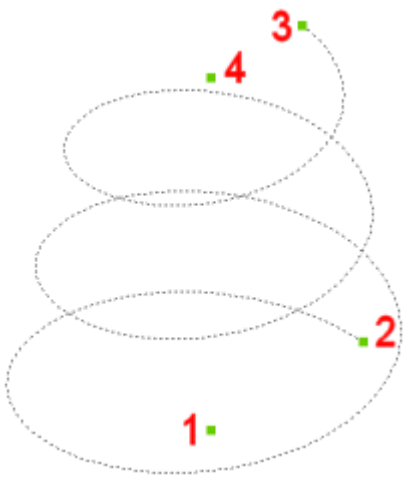
Option	Description
Diameter	Prompts you: Specify diameter <**default**>: Specify the diameter of the base of the helix.
Default	The initial default values are: • Base Radius: 1 • Top Radius: equal to the base radius • Base Diameter: 2 • Top Diameter: equal to the base diameter • Height: 1 • Turns: 3 • Twist: CCW (counter clockwise) In a drawing session, the previously used value for each parameter becomes the default value.

Axis Endpoint	Prompts you: Specify axis end point: Specify a point in the drawing. The axis endpoint defines the height and orientation of the helix.
Turns	Prompts you: Enter number of turns <**default**>: enter the number of revolutions for the helix. Numbers from 1 to 500 are accepted.
Turn Height	Prompts you: Specify distance between turns <current>: specify the height of one revolution. The turn height and the number of turns define the height of the helix.
Twist	Prompts you: Enter twist direction of helix [CW/CCW] <**default**>: type CW (clockwise) or CCW (counterclockwise).

Grips Editing

Helixes can be edited through grips:

1. Select the helix. Notice that it has four grips.
2. Drag the grips to perform the following actions:
   - (1) Move the helix.
   - (2) Modify the base radius.
   - (3) Modify the top radius.
   - (4) Modify the height.



Related Commands

3dPoly - draws 3D polylines
Pline - draws polyline lines, arcs, and splines
Spline - draws open and closed splines

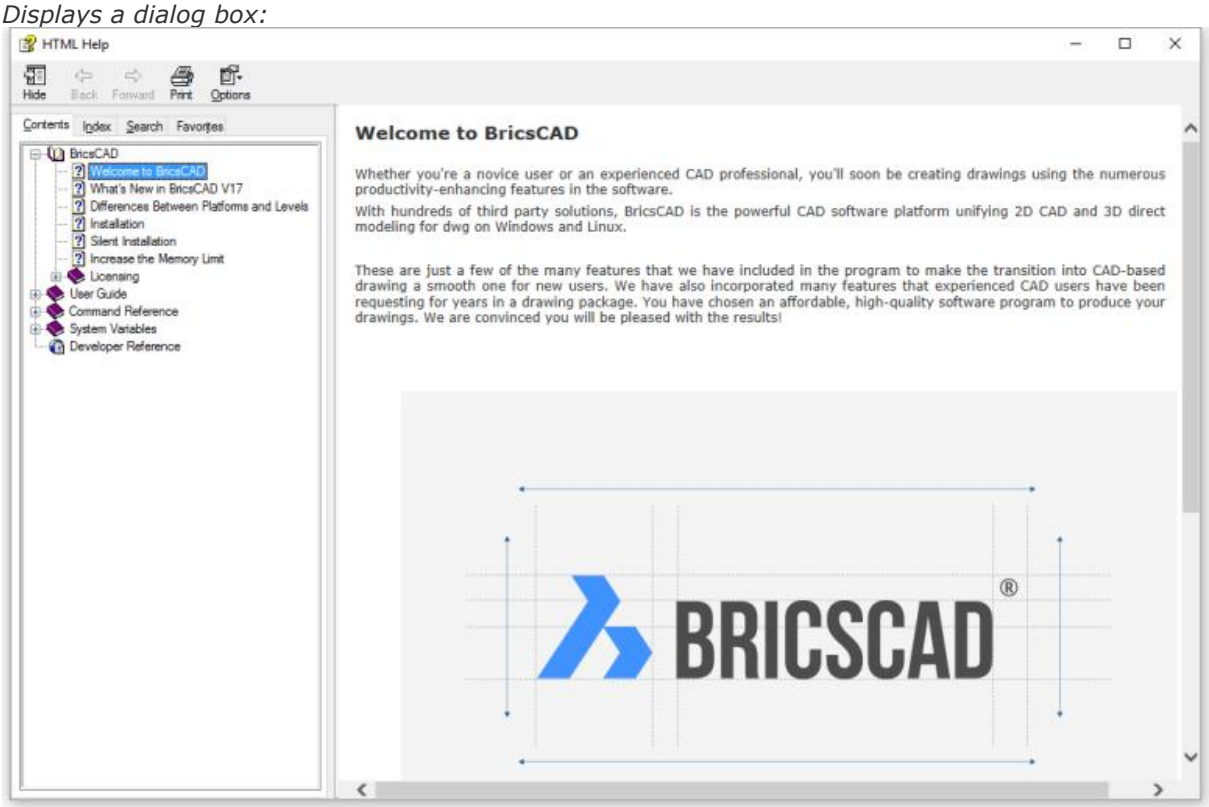
Help

BricsCAD (Windows): Opens the BricsCAD Help window.
BricsCAD (Linux): Opens the BricsCAD Online Help on the Bricsys website.
BricsCAD (Mac): Opens the BricsCAD Online Help on the Bricsys website.

Accessing the Command

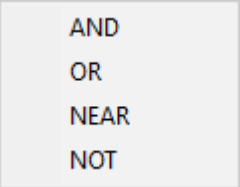
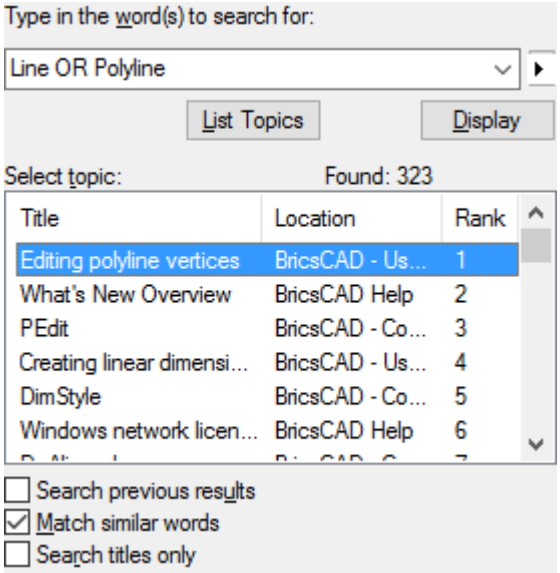
command bar: *help*
transparent: *'help*
shortcut: *F1*
alias: *?*
menu bar: *Help | Help*
toolbar: *Standard |*

: ***help***
Displays a dialog box:



Command Options

Option	Description
Contents	Lists the content of the help file in logical order.
Index	Lists the content of the help file in alphabetical order.
Search	Lists all topics which match the content of the search field: • line: lists all topics with the word 'line'. • wildcards * and ? are allowed. E.g. poly*: lists all topics containing words starting with 'poly'.

	Click the arrow button at the right hand side of the search field to add boolean operators.  • line AND polyline: lists all topics containing both words 'line' and 'polyline'. • line OR polyline: lists all topics containing the word 'line' or 'polyline'. • line NOT polyline: lists all topics containing the word 'line' but not 'polyline'. • line NEAR polyline: list all topics containing the word 'line' near 'polyline'.  Click the *List Topics* button or press Enter to start the search. Select a title and double click or click the *Display* button see the content of the topic. Click the *Title*, *Location* or *Rank* column heading to sort the topic.
Favorites	Stores topics that are accessed frequently.

Related Commands

? - opens the Help dialog box.
HelpSearch - searches the BricsCAD Help and displays a list of matching topics in a dialog box.

HelpSearch

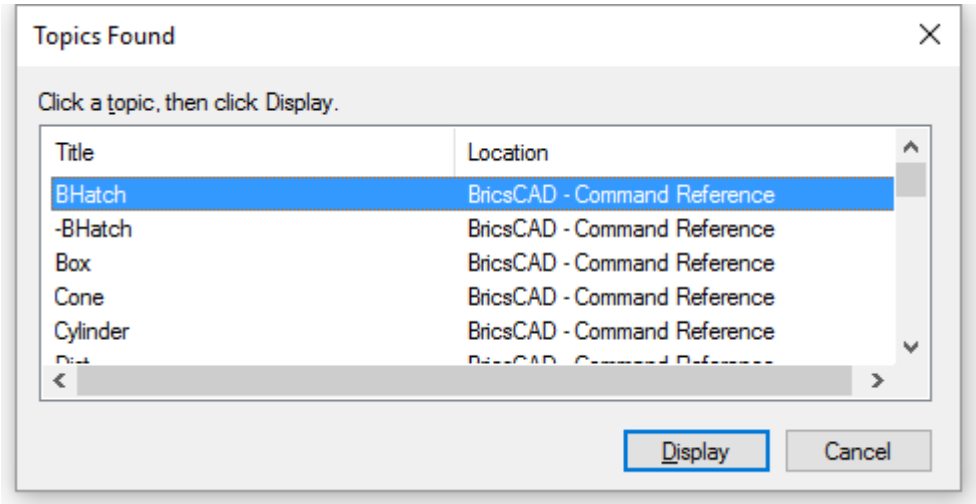
Searches the BricsCAD Help and displays a list of matching topics in a dialog box.

Accessing the Command

command bar: *helpsearch*

: ***helpsearch***
Prompts you in the command bar:
Search in Help: ***(Type a search string.)***

Displays the result in a dialog box:



Select a topic in the list, then click the *Display* button to read the selected topic in the BricsCAD Help dialog box.

Command Options

No Options

Related Commands

Help - opens the BricsCAD Help window.
? - displays the Help window.

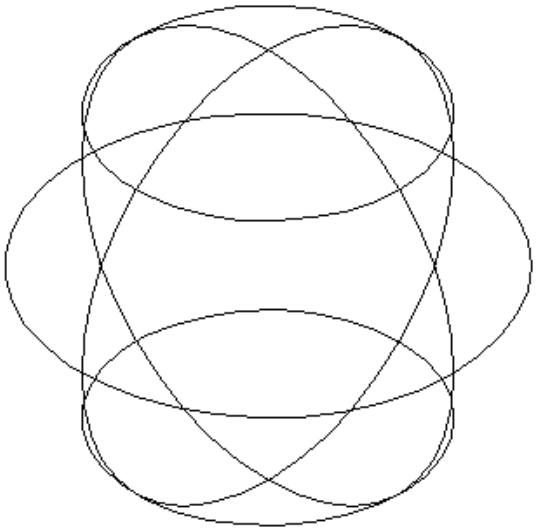
Hide

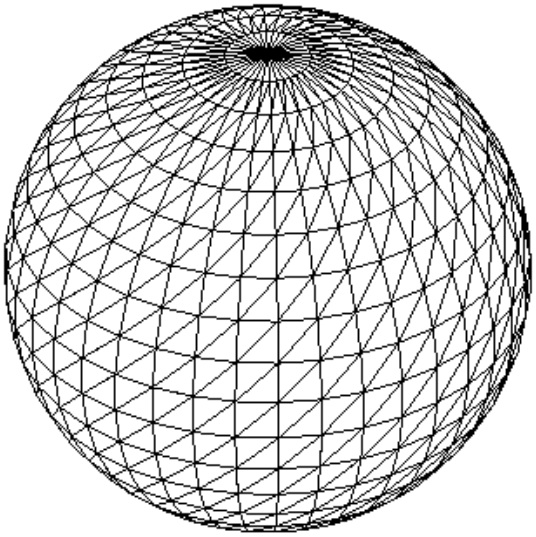
Removes hidden lines from 3D entities.

Accessing the Command

command bar: *hide*
alias: *hi*
menu bar: *View | Rendering | Hide*
toolbar: *Rendering |*

: ***hide***
No prompts in the command bar; BricsCAD immediately removes hidden lines.





Left: Wireframe representation of a 3D sphere; right: 3D sphere with hidden lines removed.

Depending on the previous shademode *2D Wireframe* or *3D wireframe* hidden lines are removed permanently or not.

To return to wireframe display:

- *use the Regen command, if 2D wireframe was the previous shademode.*
- *choose 2D Wireframe or 3D Wireframe in the Shademode command options if another shademode was active previously.*

Command Options

No options.

Related Commands

Render - generates a photorealistic rendering using materials and lights
Shade - generates shaded images of 3D drawings.
ShadeMode - sets the shading style for 3D entities.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
Flatshot - creates a hidden line representation of all 3D solids in model space as a block or a new drawing.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

VsCurrent - sets the visual style at the command line.

HideObjects

Hides the selected entities.

Accessing the Command

command bar: *hideobjects*
toolbar: *Hide/Show Entities* |
Quad: *Custom* |

: ***hideobjects***
Prompts you in the command bar:

Select Entities to Hide : (Select one or more entities.)
Entities in set: 1

Select Entities to Hide : (Select more entities or press Enter.)

The OBJECTISOLATIONMODE system variable controls whether the hidden state is saved or not:

[0] Objects are temporarily hidden for the current session. Interfered solids are selected together with the interference solids.
[1] Objects remain hidden between sessions. Interfered solids are selected together with the interference solids.
[2] Objects are temporarily hidden for the current session. Interfered solids are not selected together with the interference solids.
[3] Objects remain hidden between sessions. Interfered solids are not selected together with the interference solids.

Command Options

No options.

Related Commands

LayFrz - freezes the layers associated with entities selected in the drawing.
LayOff - turns off layers associated with entities selected in the drawing.
IsolateObjects - hides all but the selected entities.
UnisolateObjects - unhides entities that were hidden.

Hyperlink

Links entities with files and Web pages with a dialog box.

Accessing the Command

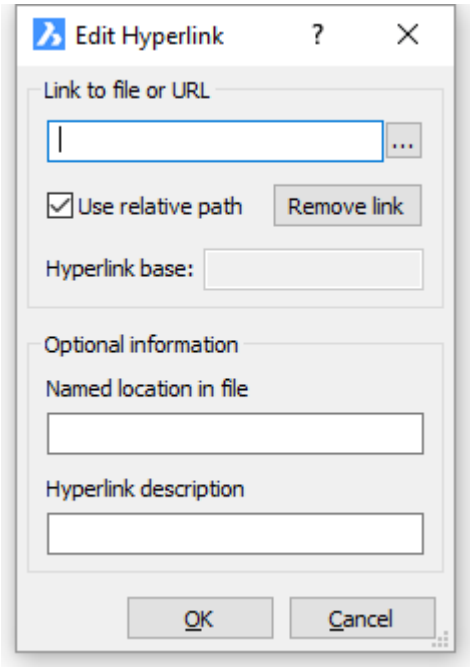
command bar: *hyperlink*
shortcut: *Ctrl+K*

: ***hyperlink***
Prompts you in the command bar:

Select entities: **(Choose one or more entities.)**
Entities in set: 1

Select entities: **(Choose one or more entities, or press Enter to continue command.)**

Displays a dialog box:



Change hyperlink options, and then click OK**.**
Each entity can contain just one hyperlink.

Command Options

Option	Description
Link to file or URL	Specifies the hyperlink; can be the path and name of a file, networked file, or location on the Internet. Locations on the Internet can include Web pages (http://), FTP sites (ftp://), and so on. Examples of hyperlink syntax are as follows: File - C:\foldername\filename.ext Networked file - \\networkname\c\foldername\filename.ext URL - https://www.bricsys.com or www.bricsys.com

...	Displays the Select File dialog box; choose a file, and then click Open. Any type of file can be linked to.
Use relative path	Uses the path specified by Hyperlink Base. Hyperlink base + relative path = full path or URL.
Remove link	Removes the link from the entity.
Hyperlink base	Specifies the path to relative hyperlinks.
Named location in file	Specifies a bookmark name. The # character is used to specify locations (bookmarks) within files or Web pages, such as www.bricscad.sys#bookmark.
Hyperlink description	Describes the URL or file; this description is displayed in a tooltip when the cursor is paused over the entity; see HyperlinkOptions command.

Related Commands

-Hyperlink - attaches hyperlinks through the command bar.
HyperlinkOptions - sets options for hyperlinks.
Settings - sets options for hyperlinks in the Files section of the Settings dialog box.
Browser and Url - open the default Web browser.
OnWeb - opens the Bricsys home page in your computer's default Web browser.
Mail - attaches the current drawing to a message in the default email client.

-Hyperlink

Links entities with files and Web pages at the command prompt.

Accessing the Command

command bar: *-hyperlink*

: ***-hyperlink***
Prompts you in the command bar:
Enter an option [Remove/Insert] <Insert>: *(Type R or I.)*

Entities can contain just one hyperlink; if you attempt to insert another hyperlink, BricsCAD asks:
Overwrite existing hyperlinks? [Yes/No] <Yes>: (Type Y or N.)

This command is meant for use by macros and LISP routines.

Command Options

Option	Description
Remove	Removes hyperlinks from entities. Prompts you: Select entities - choose one or more entities. Select entities - press Enter.
Insert	Adds hyperlinks to entities. Prompts you: Select entities - chooses the entities to which hyperlinks are to be added. **Enter URL -** specifies the path and name of the file, networked file, or location on the Internet. **Enter named location** - specifies a bookmark name; press Enter to ignore this option. Enter description - describes the hyperlink; press Enter to leave empty.

Overwrite Option	Description
Yes	Replaces the existing hyperlink with the new one.
No	Leaves the existing hyperlink; does not replace the existing one.

See the **Hyperlink** command for example URLs.

Related Commands

Hyperlink - attaches hyperlinks through a dialog box.
HyperlinkOptions - sets options for hyperlinks.
Settings - sets options for hyperlinks in the Files section of the Settings dialog box.
Browser and Url - open the default Web browser.
OnWeb - opens the Bricsys home page in your computer's default Web browser.
Mail - attaches the current drawing to a message in the default email client.

HyperlinkOptions

Toggles the display of the hyperlink cursor, shortcut menu, and tooltips.

Accessing the Command

command bar: *hyperlinkoptions*
transparent: *'hyperlinkoptions*

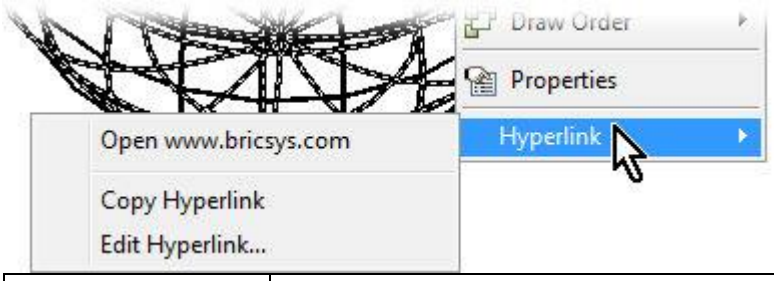
: ***hyperlinkoptions***
Prompts you in the command bar:

Display hyperlink cursor and shortcut menu? [Yes/No] <Yes>: (Type Y or N.)

Display hyperlink tooltips? [Yes/No] <Yes>: (Type Y or N.)

Command Options

Option	Description
Display hyperlink cursor and shortcut menu	Toggles the display of the "hyperlink" icon and the Hyperlink submenu on shortcut menus: • Yes - turns on the display. • No - turns off the display. When the cursor passes over entities with hyperlinks, the "world" icon is displayed. When you right-click a selected entity that contains hyperlinks, the Hyperlink item is added to the menu.
Display hyperlink tooltips	Toggles the display of the hyperlink tooltip: • Yes - turns on the display. • No - turns off the display. The text of the tooltip is specified by the **Hyperlink** command's Hyperlink Description option.



Hyperlink Cursor	Description
Open *url*	Opens the location specified by the URL. If a location on the Internet, BricsCAD opens the page in the default Web browser.

	If a file, BricsCAD opens the file in the associated application. This option executes the URL command. See URL command.
Copy Hyperlink	Copies the URL to the Clipboard; the URL can be pasted into the drawing or other documents with the Ctrl+V shortcut. See PasteClip command.
Edit Hyperlink	Opens the Edit Hyperlink dialog box for editing the URL, description, and other parameters. See Hyperlink command.

Related Commands

-Hyperlink - attaches hyperlinks through the command bar.
Hyperlink - attaches hyperlinks through a dialog box.
Settings - sets options for hyperlinks in the Files section of the Settings dialog box.
Browser and Url - open the default Web browser.
OnWeb - opens the Bricsys home page in your computer's default Web browser.
Mail - attaches the current drawing to a message in the default email client.

I

Id

Reports the x,y,z coordinates of picked points in the current coordinate system.

Accessing the Command

command bar: *id*
transparent: *'id*
shortcut: *Ctrl*
alias: *idpoint*
menu bar: *Tools | Inquiry | ID Coordinates*
toolbar: *Inquiry |*

: ***id***
Prompts you in the command bar:

Select point to identify coordinates: *(Pick a point in the drawing.)*

X=17.9824 Y-2.7442 Z=0.0000

Use entity snaps to pick points accurately on geometric features.

Command Options

Option	Description
Select point to identify coordinates	Specifies the point to be reported.

Related Commands

Area - reports the area, circumference, and perimeter of entities.
Dist - reports the 2D and 3D distances between two points.
List - lists the area and other parameters of entities in the command bar.
MassProp - reports the properties of solid models and regions.
Properties - lists the area and other parameters of entities in a palette.

Image

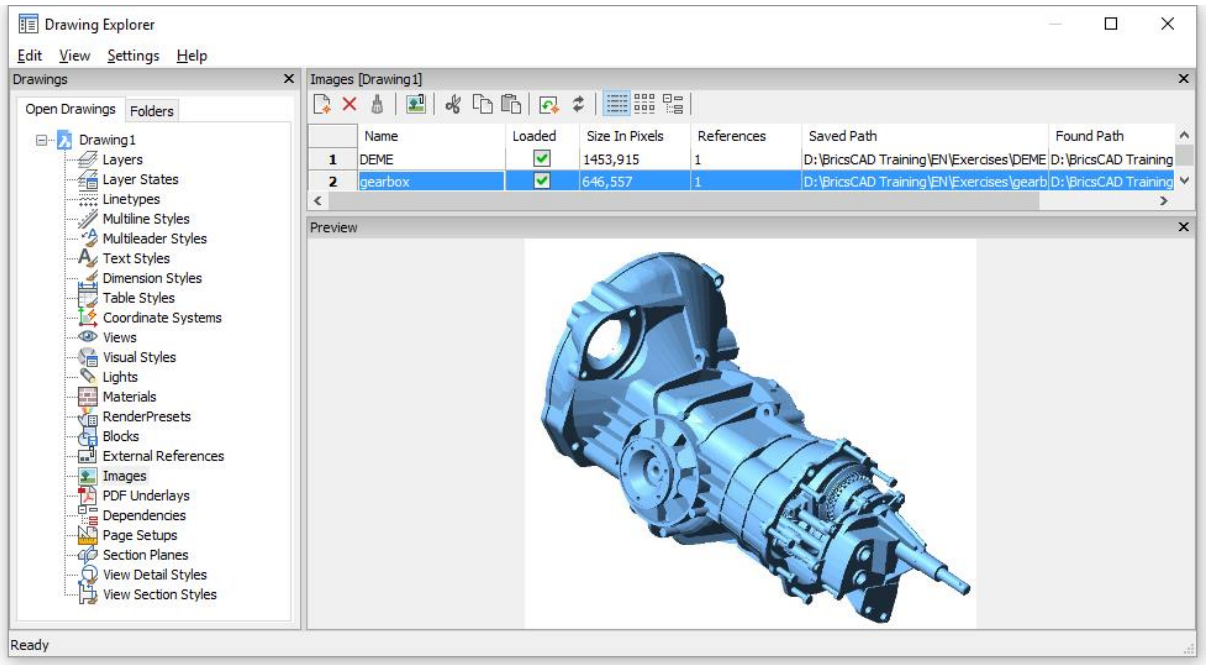
Inserts images in drawings through the Drawing Explorer.

Accessing the Command

command bar: *image*
aliases: *expimages, im*
menu bar: *Tools | Drawing Explorer | Images*
toolbar: *Images |*

: ***image***

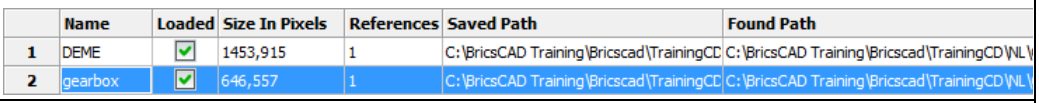
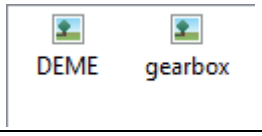
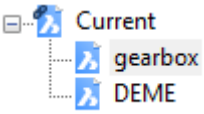
Displays the Drawing Explorer open to the Images node:



*Attach image files, or modify their properties,and then click **X**.*
Press F1 to access help.

Command Options

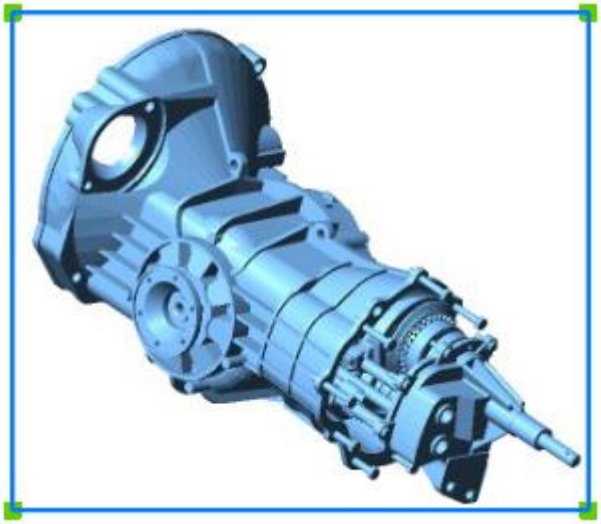
Option	Description
New	Attaches raster image files to the drawing; displays the Open dialog box. Choose a raster file, and then click Open. BricsCAD displays the Images dialog box. See **ImageAttach** command.
Delete	Erases the image from the drawing. You may need to use the **Regen** command to update the screen.
Insert	Inserts the image in the drawing; displays the Images dialog box. See **ImageAttach** command.
Cut	Cuts the image to the Clipboard. You can paste it in the Drawing Explorer's Image section of another drawing. It is removed from the current drawing.

Copy	Copies the image to the Clipboard. You can paste it in the Drawing Explorer's Image section of another drawing.
Paste	Pastes images and related data.
Regen	Updates the drawing to reflect changes made in this dialog box.
Regen Toggle	Toggles automatic drawing updates: • On - updates drawing automatically. • Off - updates drawing only when the Redraw button is clicked.
Details	Displays the details of each image.  Name Loaded Size In Pixels References Saved Path Found Path 1 DEME 1453,915 1 C:\BricsCAD Training\Bricscad\TrainingCD C:\BricsCAD Training\Bricscad\TrainingCD\NL\ 2 gearbox 646,557 1 C:\BricsCAD Training\Bricscad\TrainingCD C:\BricsCAD Training\Bricscad\TrainingCD\NL\
Icons	Displays the image files as small icons, hiding their details. This is useful when a drawing contains many images.  DEME gearbox
Tree	Displays the relationship between files in an Explorer tree-like fashion. "Current" refers to the current drawing.  Current gearbox DEME
Name	Specifies the name of the image. This is usually the image's file name.
Loaded	Toggles whether the image is displayed: • On - displays the image in the drawing. • Off - does not display the image in the drawing; the image frame is visible, unless 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	Specifies the path to the image file. Click the ... button to change the path.
Found Path	Reports the path to the image file.

Grips Editing

Images can be edited directly through grips:

1. Select the image. Notice that its frame has four grips.
2. Drag a grip to stretch (resize) the image.



Procedures

Open the Images Explorer
Images Explorer display options
Placing images in a drawing
To load / Unload images
Detaching images

Related Commands

ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageAdjust - changes the properties of images through the Properties palette.
ImageClip - clips images.
ImageFrame - toggles the display of the frame around images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

Explorer - Opens the Drawing Explorer dialog box.

ImageAdjust

Adjusts the properties of images through the Properties palette.

Accessing the Command

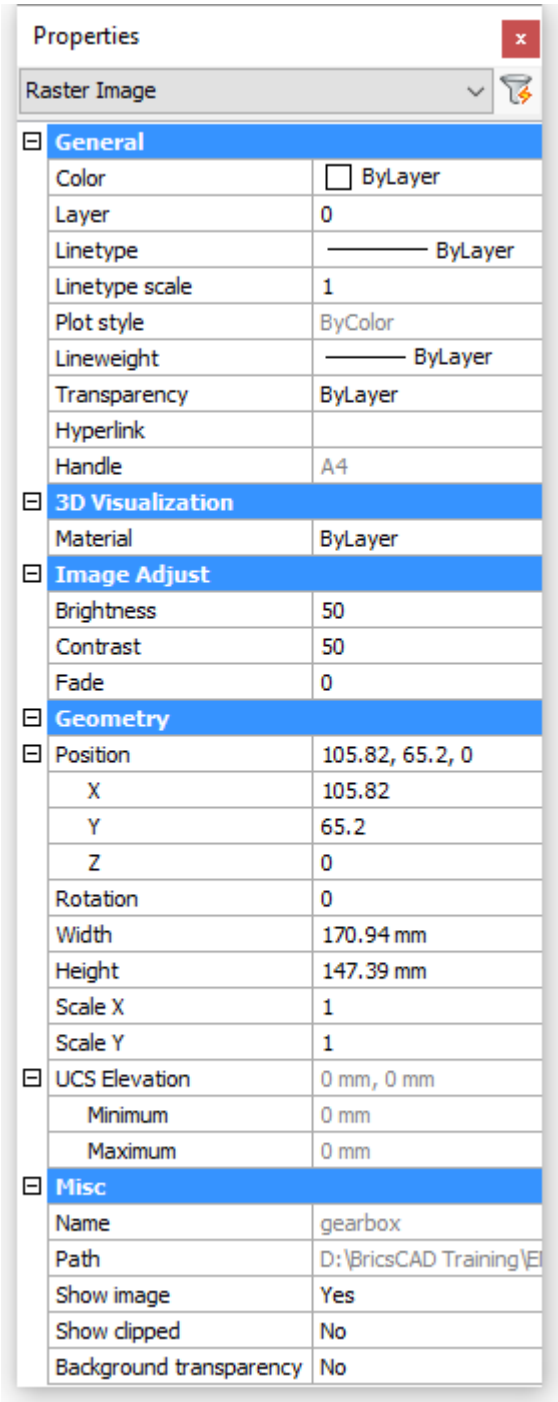
command bar: *imageadjust*
transparent: *'imageadjust*
shortcut: *double-click the image frame*
alias: *iad*
toolbar: *Images* |

: ***imageadjust***
Prompts you in the command bar:

Select entities: (Choose one or more image entities.)

Select entities: (Press Enter.)

Displays the properties palette:



Change properties.

Command Options

Option	Description
Select entities	Selects the images to be adjusted.

General Properties	*The following properties affect the frame that surrounds the image:*
Color	Specifies the color of the frame. See Color command.
Layer	Specifies the layer of the frame. When the image is placed on its own layer, you can use layer visibility to toggle the display of the image and its frame. Use the ImageFrame command to toggle the display of the frame independently of the image. See Layer command.
Linetype	Specifies the linetype of the frame. See Linetype command.
Linetype Scale	Specifies the linetype scale of the frame. See LTSCALE system variable.
Plot Style	Specifies the plot style of the frame. This option is available only when plot styles are enabled in the drawing, See PlotStyle command.
Lineweight	Specifies the lineweight of the frame. See LWeight command.
Hyperlink	Specifies a URL or path to a file. See Hyperlink command.
Image Adjust	*The following properties affect the image:*
Brightness	Lightens and darkens the image: • 0 - very dark or black. • 50- normal. • 100 - very light or white.
Contrast	Increases and decreases the contrast: • 0 - very low contrast. • 50- normal. • 100 - very high contrast.
Fade	Fades the image: • 0 - no fade. • 100 - fully faded.
Geometry	*The following properties affect the image:*
Position	Specifies the x, y, and z coordinates of the image's lower right corner. When you change the coordinates, the lower right corner is marked with a large X.
Rotation	Specifies the rotation angle about the position point: • Positive angles - rotate the image counter-clockwise. • Negative angles - rotate in the image clockwise.

Width	Specifies the width of the image, in current units.
Height	Specifies the height of the image, in current units.
Scale	Specifies the scale factor of the image: • Less than 1 - makes the image smaller. • More than 1 - makes the image larger.
Misc	
Name	Reports the name of the image; cannot be edited by the user.
Path	Reports the path to the image file; cannot be edited by the user.
Show Image	Toggles display of the image: • Yes - shows the image; this can be overridden by setting the layer to Frozen. • No - hides the image.
Show Clipped	Toggles clipping: • Yes - displays clipped image, if clipping has been previously applied by ImageClip command; part of the image is hidden. • No - hides clipped portion(s); the full image is displayed. See ImageClip command.
Background Transparency	Toggles the background transparency: • Yes - makes the background color of the image. • No - keeps the image nontransparent. *Transparency* makes one color of the image see-through. See Transparency command.

Related Commands

Image - controls the parameters of the image file.
ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageClip - clips images.
ImageFrame - toggles the display of the frame around images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

ImageAttach

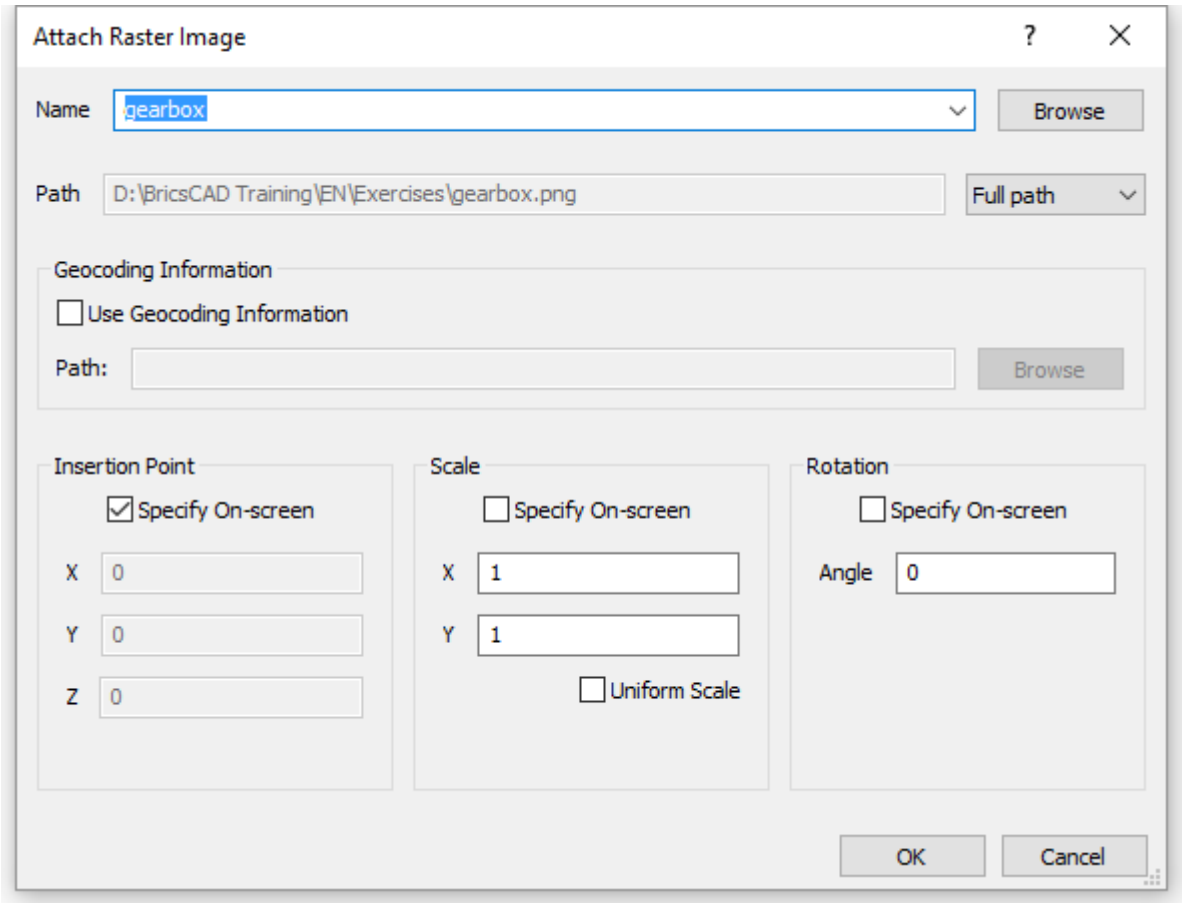
Attaches raster images to the drawing; displays a dialog box.

Accessing the Command

command bar: *imageattach*
transparent: *'imageattach*
alias: *iat*
toolbar: *Images* | and *Insert* |

: ***imageattach***

Displays a dialog box:



Specify options, and then click ***OK****.*

This command inserts the following file formats into drawings:

BMP - Windows bitmap.

JPG, JPEG and JPEG2000 (JP2, j2k) - Joint Photographic Experts Group (common among digital cameras).

PCX - PC Paintbrush.

PNG - Portable network graphics.

GIF - CompuServe graphic image format.

TGA - Targa.

TIF - Tagged image format (commonly used by desktop publishing).

ECW - Enhanced Compressed Wavelet format.

SID - MrSID is an acronym for Multi-resolution Seamless Image Database.

Command Options

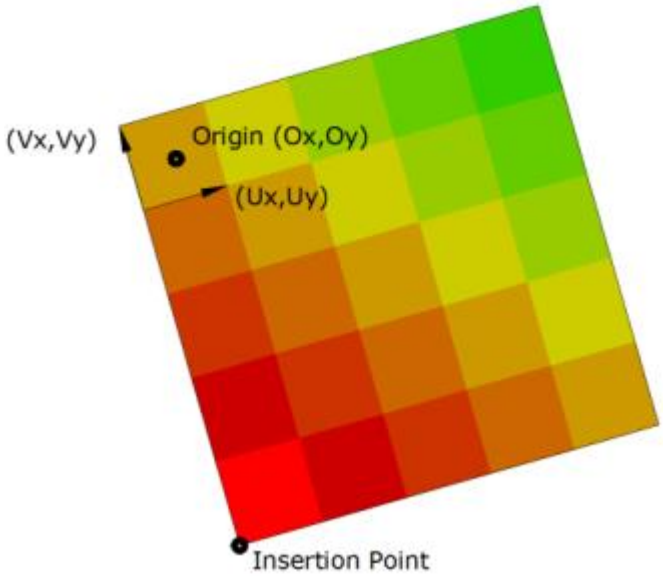
Options	Description
Name	Specifies the file name of the image.
Browse	Displays the Open dialog box; click this button to choose a different raster file. Choose a raster file, and then click Open.
Path	Reports the path to the image file; cannot be edited by the user.
Full Path	Determines how much of the path is stored: • Full Path - stores the full path to the file, including drive and folder names. For example: c:\foldername\imagename.ext • Relative Path - stores the path from the drawing's location to the image's location; the drawing must be saved before this option can be selected. For example: ..\foldername\imagename.ext where .. refers to the folder above the current one. • No path - the path is discarded.
Geocoding Information	Position files or world files have an extension that depends on what kind of image file they are associated with. See explanatory notes below: "About Geocoding Information".
Use Geocoding Information	Toggles use of a position file, which specifies the size, location, and rotation of the image file: • Yes - uses a position file. • No - does not use a position file.
Path	Reports the path to the position file.
Browse	Displays the Select Image Position File dialog box; choose a TWF file, and then click Open.
Insertion Point	Positions the lower left corner of the image in the drawing. This option is unavailable when a position file is used.
Specify On-screen	Toggles where you specify the insertion coordinates: • Off - Specifies the x,y,z coordinates in the dialog box. • On - Specifies the coordinates after you click OK, in the command bar: Insertion point - specify the x,y or x,y,z coordinates of the image's lower left corner.
XYZ	Specifies the x-, y- and z-coordinate of the image's lower left corner.

Scale	Sizes the image. This option is unavailable when a position file is used.

Specify On-screen	Toggles where you specify the scale factors: • Off - Specifies the x,y,z scales in the dialog box. • On - Specifies the scales in the command bar after you click OK: Scale factor - specify the scale factor.
XY	Specifies the scale factor in the x and y direction: • Values larger than 1 make the image larger. • 1 inserts the image at actual size. • Values smaller than 1 make the image smaller.
Uniform Scale	Makes the Y scale factor equal to X.
Rotation	Rotates the image about the insertion point.
Specify On-screen	Toggles where you specify the rotation angle: • Off - Specifies the angle in the dialog box. • On - Specifies the angle in the command bar after you click OK:. Prompts you: Rotation angle - specify the angle.
Angle	Specifies the rotation angle about the insertion point. • Positive angles rotate the image counterclockwise. • Negative angles rotate the image clockwise.

About Geocoding Information

Position files (or world files) are plain ASCII text files that position raster files in the drawing. They are useful for positioning images according to geographic coordinates, such as latitude and longitude. They can be created using text editors such as *Notepad*.
The format is a list of 6 numbers that fully describe location, scale and orientation of the top left pixel of the raster file to be attached.
If *U* is the vector along the width, *V* is the vector along the height, and *O* is the center of the pixel (see image below):


(Vx,Vy)
Origin (Ox,Oy)
(Ux,Uy)
Insertion Point

then the list consists of:
Ux
Uy
Vx
Vy
Ox
Oy

Typically, the first and last letters of the image extension are used with a ”w” added at the end.
Sometimes the full image extension with a ”w” at the end is used also.
The following position file formats are supported:

- *.bpw (for BMP images)
- *.jgw (for JPG images)
- *.gfw (for GIF images)
- *.tfw (for TIFF images)
- *.j2w (for JPEG2000 images)
- *.sdw (for SID images)
- *.pgw (for PNG images)

Procedures

Placing images in a drawing

Related Commands

-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
Image - controls the parameters of the image file.
ImageAdjust - changes the properties of images through the Properties palette.
ImageClip - clips images.
ImageFrame - toggles the display of the frame around images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

-ImageAttach

Attaches raster images to the drawing; displays prompts in the command bar.

Accessing the Command

command bar: *-imageattach*

: ***-imageattach***
Prompts you in the command bar:

Image path will be stored as Full Path.

Set image path mode [Full path/Relative path/No path] <Image path>: (Type the path and name of the image file in the command bar or choose a path option.)

Use geocoding information [use Geocoding info] <Insertion point>: (Pick a point or choose an option.)

Scale [Size/XY scales] <1.000000>: (Pick a point, press Enter to accept the default scale, type a value or choose an option.)

Rotation angle <0>: (Press Enter to accept the default rotation angle, pick a point or type a value.)

Command Options

Option	Description
Image Path	Allows to choose how the image path is saved in the drawing. The options are: • Full Path - the full path of the image file is saved; e.g. D:\BricsCAD Training\EN\Exercises\Gearbox.png • Relative Path - the relative path with respect to the drawing folder is saved; e.g. ..\Exercises\Gearbox.png • No Path - BricsCAD searches for the image in the drawing folder.
Use geocoding information	Allows to use geocoding information to insert the image. Prompts you: **Embedded geocoding information/Geocoding file path [Embedded]<Path\ImageName.pgw*>:** • Press Enter to accept the position file name. • Embedded - Use embedded geocoding information * See: About Geocoding Information.
XY Scales	Allows to set the scaling in X- and Y direction differently. Prompts you: Specify opposite corner or X scale factor <1.000000>: Do one of the following: • Specify a point in the drawing. • Type a value.
Size	Sets the size of the image. Prompts you: Scale [Size/XY scales] <1.000000> - Pick a point or specify the XY scale factor; prompts you: Specify X scale factor <1.000000> - Type a value.

	Specify Y scale factor < Equal to X scale factor <1.000000> - Press Enter to set the Y scale factor equal to the X scale factor or type a value.

Rotation Angle	Specifies the rotation angle. Prompts you: Rotation angle <0> - Press Enter to accept the default 0° rotation, or pick a point or type a value.

Related Commands

ImageAttach - attaches raster images to the drawing; displays a dialog box.
Image - controls the parameters of the image file.
ImageAdjust - changes the properties of images through the Properties palette.
ImageClip - clips images.
ImageFrame - toggles the display of the frame around images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

ImageClip

Clips images.

Accessing the Command

command bar: *imageclip*
transparent: *'imageclip*
alias: *icl*
menu bar: *Modify | Raster Image | Clip Image*
toolbar: *Images |*

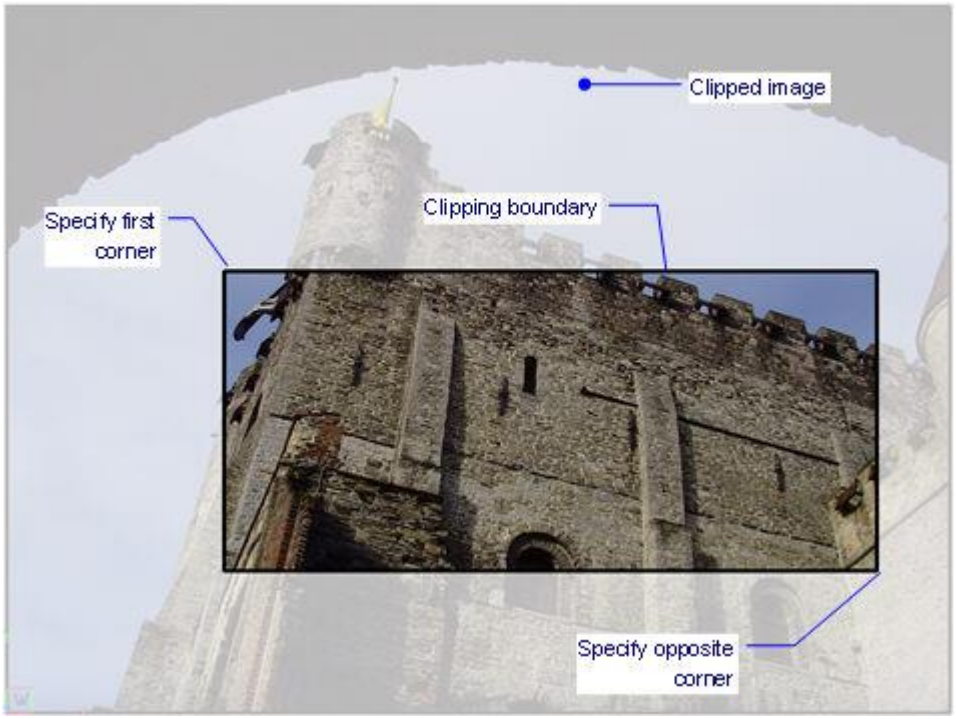
: ***imageclip***
Prompts you in the command bar:

Select image: (Select an image.)

Enter image clipping option [ON/OFf/Invert/Delete boundary/New boundary] <New boundary>: (Enter an option, or press Enter to create a new clipping boundary.)

Enter image clipping type [Polygonal/Rectangular] <Rectangular>: (Press Enter to create a rectangular clipping boundary or choose Polygonal.)
Specify first corner: (Pick one corner of the clipping rectangle.)

Specify opposite corner: (Pick other corner of the clipping rectangle.)



This figure shows the clipped portion faded for illustrative purposes; in BricsCAD, the clipped portion is invisible.

An image can have just one clipping boundary; creating a new one erases the old one.

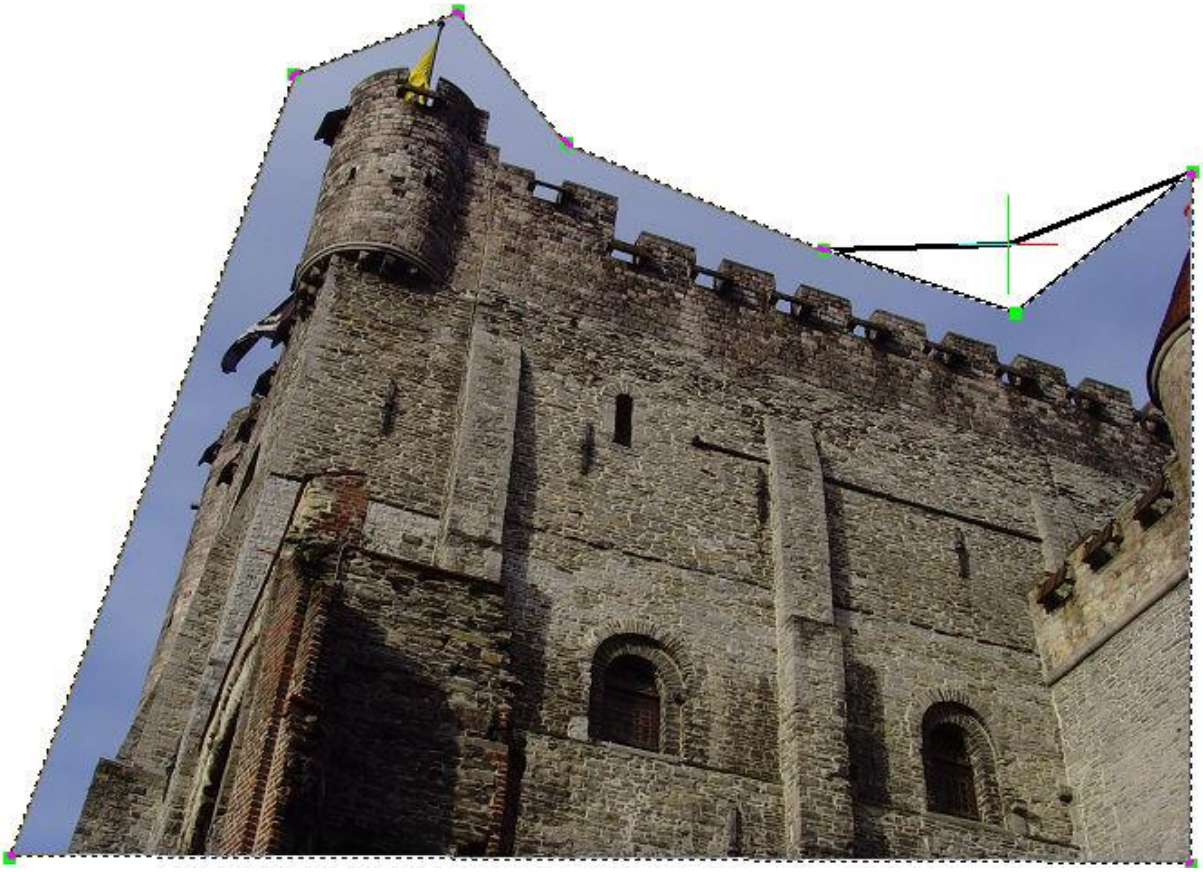
Command Options

Option	Description
Enter clipping option	Determines the clipping action: • ON - turns on the clipping and displays the clipping boundary. • OFF - turns off the clipping and hides the clipping boundary. • Delete Boundary - removes the clipping boundary. • New Boundary - creates a new clipping boundary, and turns on clipping. • Invert - inverts he clipping mode: selected images are clipped either outside or inside the clipping boundary.
Rectangular	Creates a rectangular clipping boundary; prompts you: **Specify first corner** - specifies the first corner of the clipping rectangle. Specify opposite corner - specifies the other corner of the clipping rectangle.
Polygonal	Creates a multi-sided, polygonal boundary; prompts you: Specify first point - specifies the starting point of the clipping boundary. Specify next point or [Undo] - specifies the next point; as you pick points, BricsCAD previews the boundary. Specify next point or [Undo] - specifies the next point, or undoes the last one; this prompt repeats until you press Enter.

Grips Editing

Clipped images can be edited directly through grips:

1. Select the image frame. Notice that it has several grips.
2. Drag a grip to stretch the frame, which changes the path of the clipping boundary.



Procedures

To clip an image
To toggle the clipping boundary of an image

Related Commands

Image - controls the parameters of the image file.
ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageAdjust - changes the properties of images through the Properties palette.
ImageFrame - toggles the display of the frame around images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

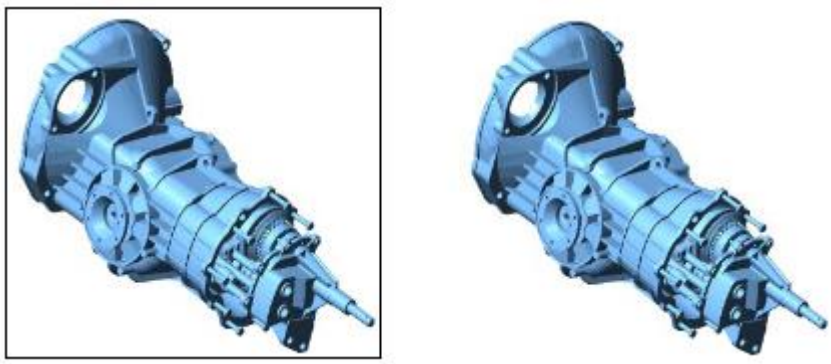
ImageFrame

Toggles the frame around images.

Accessing the Command

command bar: *imageframe*
transparent: *'imageframe*
menu bar: *Modify | Raster Image | Display Image Frame*
toolbar: *Images |*

: ***imageframe***
Prompts you in the command bar:
Enter image frame setting <0>: *(Type 0 or 1)*



Left: Image frame on; right: image frame off.

This command applies to all images in the drawing; frames cannot be toggled on individual images. It also applies to the frames of rectangular and polygonal clippings.
When frames are turned off, images can only be selected using window selection.

Command Options

Option	Description
Enter image frame setting	Toggles the display of image frames: • 1 - turns on image frames. • 0 - turns off image frames.

Related Commands

Image - controls the parameters of the image file.
ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageAdjust - changes the properties of images through the Properties palette.
ImageClip - clips images.
ImageQuality - toggles the quality of the image.
Transparency - toggles the transparency of monotone images.

ImageQuality

Toggles the quality of images between draft and high.

Accessing the Command

command bar: *imagequality*
transparent: *'imagequality*
menu bar: *Modify | Raster Image | Image Quality*
toolbar: *Images |*

: ***imagequality***
Prompts you in the command bar:
Enter image quality setting [Draft/High] <High>: *(Type D or H.)*

image quality = high

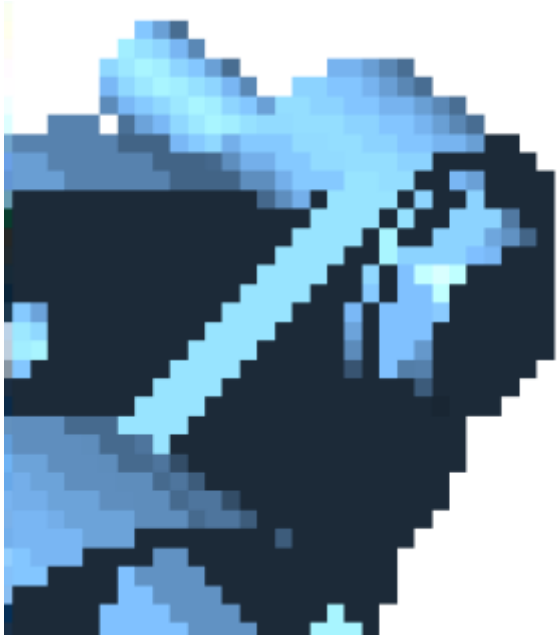


image quality = draft

Use this command to reduce the memory used by very large images.
When set to Draft, this command:

- Reduces color resolution.
- Scales images with large numbers of pixels to reduce resolution and memory footprint.

Command Options

Option	Description
Enter image quality setting	Toggles the image display quality: • Draft - lowers quality, displays faster. • High - raises quality, displays slower.

Related Commands

Image - controls the parameters of the image file.
ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageAdjust - changes the properties of images through the Properties palette.
ImageClip - clips images.
ImageFrame - toggles the display of the frame around images.
Transparency - toggles the transparency of monotone images.

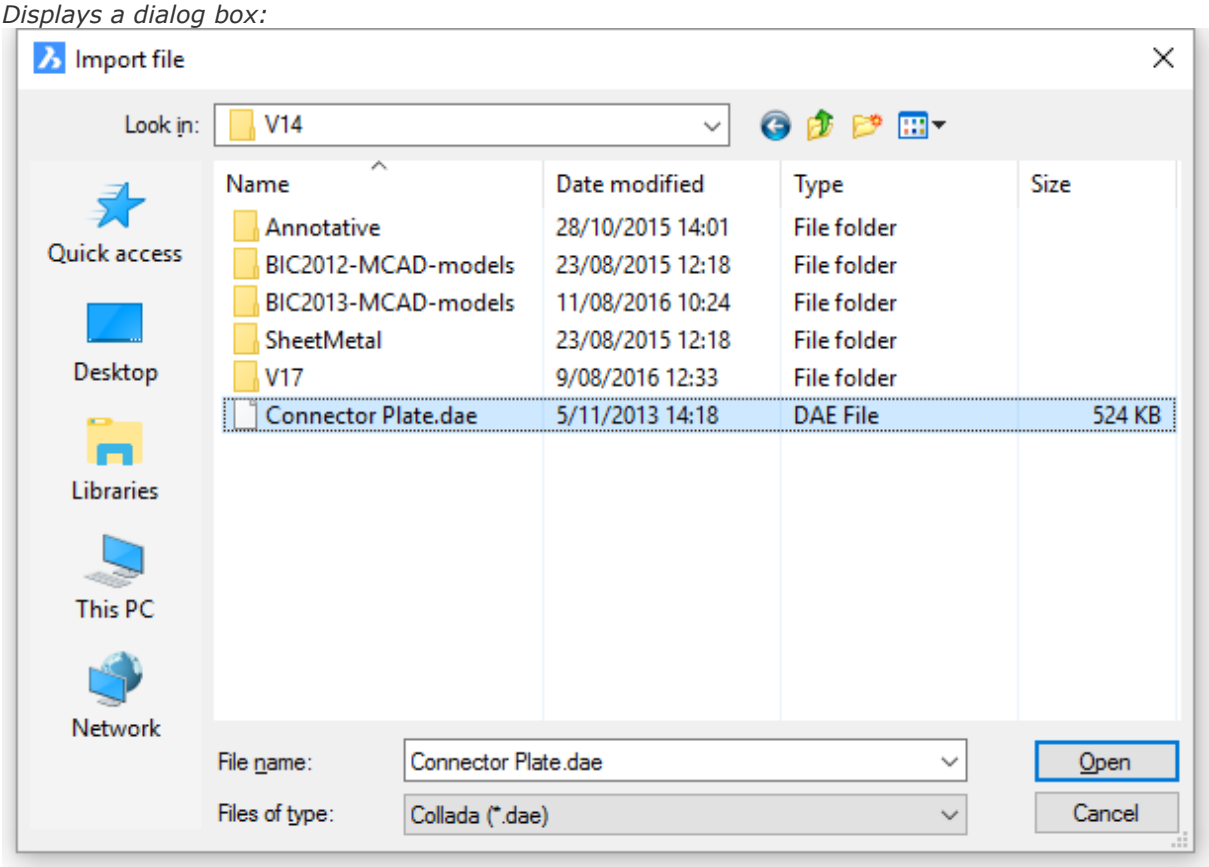
Import

Imports the geometry from external files into the current drawing.

Accessing the Command

command bar: *import*
menu bar: *File | Import*

: ***import***
Displays a dialog box:



Choose a file, and then click Open.

Command Options

Option	Description
File name	Specifies the name of the file.
Files of type	Specifies the file type. All levels: • DWG - drawing files. • DXF - drawing interchange files. • DWT - template files. • DAE - collada files BIM only:

	• IFC - The Industry Foundation Classes (IFC) data model describes building and construction industry data.
	Following options are available in the Settings dialog:
	  • Import spaces: When checked, spaces are imported as 3D Solids.
	  • Import Bim Data: When checked, imported objects are classified as Building Elements, Walls, Slabs, Beams, Columns, Windows or Doors according to their IFC class.
	• SKP - Sketchup model files
	Depending on the value of the SkpStitch user preference:
	  • With stitching, faces are combined to form 3d solids, when the face edges are coincident.
	  • Without stitching, all faces are imported as REGION entities, which have layer and color properties of the corresponding Sketchup faces.

Open	Inserts the geometry in the current drawing.

The following 3D formats are available through a separate module, BricsCAD Communicator.
The ImportProductStructure user preference controls whether the structure is imported as regular blocks or as local mechanical components (Platinum license required).
The ImportRepair, ImportStitch and ImportSimplify user preferences control if imported geometry should be automatically repaired, stitched and simplified.
Download the *Communicator* installer from the Bricsys website.

Format	**Extension**	**Description**
Standard for Exchange of Product data	STE, STP, STEP	STEP (Standard for the Exchange of Product Data) is an ISO standard exchange format used for representing three-dimensional data in a format that can be recognized by multiple programs.
Initial Graphics Exchange Specification	IGS, IGES	A file format which defines a vendor neutral data format that allows the digital exchange of information among Computer-aided design (CAD) systems. The ImportIgesStitch and ImportIgesSimplify user preferences control if geometry imported from IGES/IGS files should be automatically stitched and simplified.
Inventor	IAM, IPT	Autodesk Inventor, developed by U.S.-based software company Autodesk, is 3D mechanical CAD design software for creating 3D digital prototypes used in the design, visualization and simulation of products. Inventor uses specific file formats for parts (IPT) and assemblies (IAM).
JT	JT	JT is a 3D data format developed by Siemens PLM Software (formerly UGS Corp.) and is used for product visualization, collaboration, and CAD data exchange. It can contain any combination of approximate (faceted) data, boundary representation surfaces (NURBS), Product and Manufacturing Information (PMI), and Metadata (textual attributes) either exported from the native CAD system or inserted by a product data management (PDM) system.
CATIA V4	model	CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes.
CATIA V5	CATPart, CATProduct	
PARASOLID	XMT_TXT, XMT_BIN	Parasolid is a geometric modeling kernel originally developed by ShapeData, now owned by Siemens PLM Software (formerly

	X_T, X_B	UGS Corp.), that can be licensed by other companies for use in their 3D computer graphics software products.
Solid Edge	PAR, PSM	Solid Edge is a 3D CAD parametric feature solid modeling software. which provides solid modeling, assembly modelling and drafting functionality.
NX	PRT	PRT is a part or model file format used by PTC Pro/Engineer, which is a CAD program that creates design parts and assembly processes.
Creo Elements Pro Engineer	ASM, PRT	Parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC).
Solidworks	SLDASM, SLDPRT	SolidWorks is a 3D mechanical CAD program that runs on Microsoft Windows and is being developed by Dassault Systèmes. SLDPRT: part file, SLDASM: assembly file The ImportSwRotateYZ user preference controls if geometry imported from SolidWorks files should be automatically rotated to map Y axis to Z axis.
VDA-FS	VDA	VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name is an abbreviation of "Verband der Automobilindustrie - Flächenschnittstelle", which translates to the "automotive industry association - surface data interface". Standard was specified by the German organization VDA.

Physical materials

If the imported assembly has physical materials assigned to its parts, the materials can be imported together with the assembly structure. For each material in the imported file, a corresponding material will be created in the target document material library, and its name, density, specific heat and thermal conductivity will be copied from the source material. If the imported file contains several materials with the same name, the first one will be used. If the document already has a material with the same name in its material library, this material will be used instead.

Materials copied to the document material library will not be erased in Undo operations.

Related Commands

CommunicatorInfo - checks whether the Communicator product is correctly installed and prints a diagnostic report in the Prompt History window.
dmSimplify - simplifies the geometry and topology of 3d solid entities.
dmStitch - converts a set of region and surface entities that bound a watertight area to a 3d solid.
dmAudit - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).
Export - exports the current drawing in a variety of raster and vector formats.
Insert - inserts DWG, DXF, and DWT files.
InsertObj - inserts objects from other applications.
Load - loads SHX shape files.
VSlide - displays SLD, WMF, and EMF files.

Imprint

Imprints 2D entities onto planar faces of 3D solids and surfaces; allows to create additional edges on planar faces.

Accessing the Command

command bar: *imprint*
menu bar: *Model | 3D Solid Editing | Imprint*
toolbar: *3D Solid Editing |*

: ***imprint***
Prompts you in the command bar:

Select 3d solid, surface or region: (Select a 3D solid, surface or region entity.)

Select an entity to imprint: (Select a 2D entity lying on or intersecting a planar face of the selected entity.)

Delete the source object Yes/<No>: (Choose an option.)

Select an entity to imprint: (Select another 2D entity or press Enter to stop.)

Command Options

No command options.

Related Commands

ProjectGeometry - projects 2D geometry onto a region, a surface or a 3D solid; allows to create additional linework on the faces of solids and surfaces.
SolidEdit - edits 3D solids and 2D regions.
XEdges - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Insert

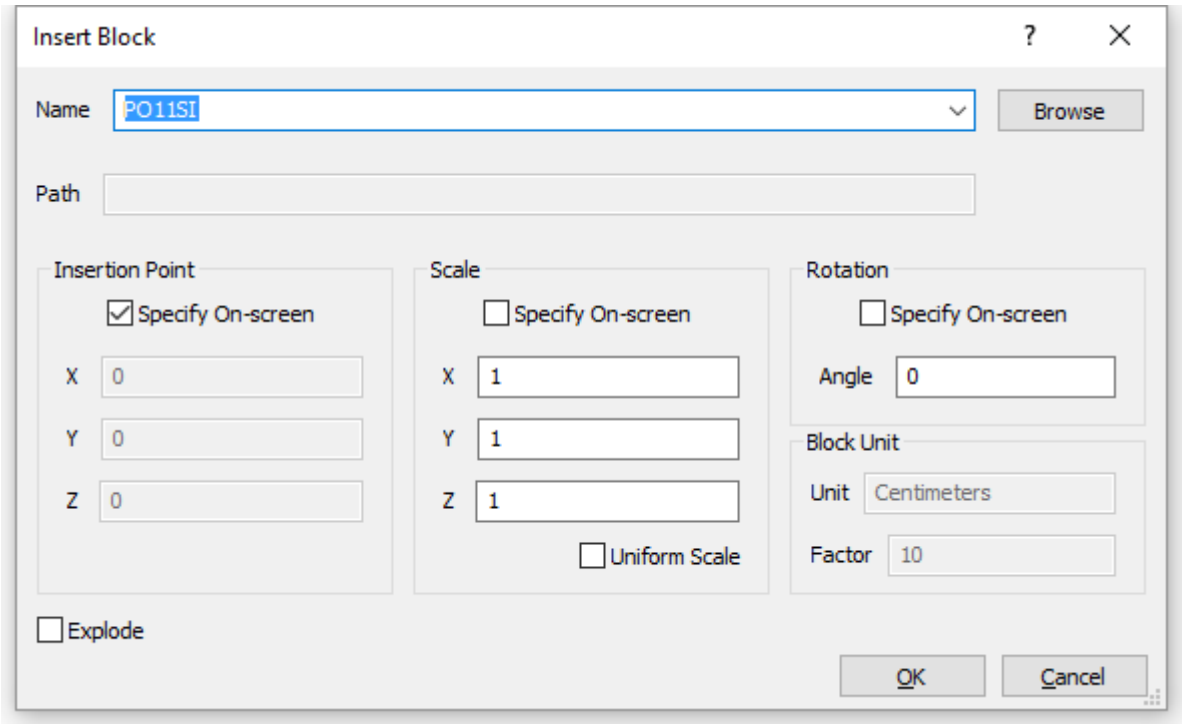
Inserts blocks, through a dialog box and prompts for attribute values through the command bar or a dialog box.

Accessing the Command

command bar: *insert*
aliases: *i, ddinsert*
menu bar: *Insert | Insert Block*
toolbar: *Insert |*

: ***insert***

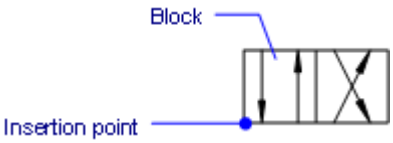
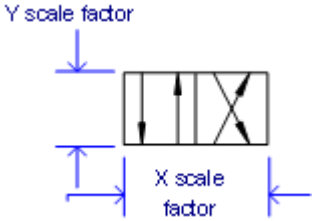
Displays a dialog box:

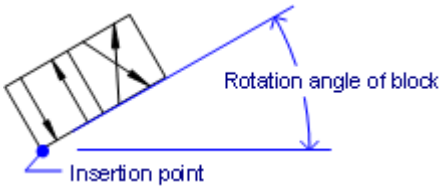
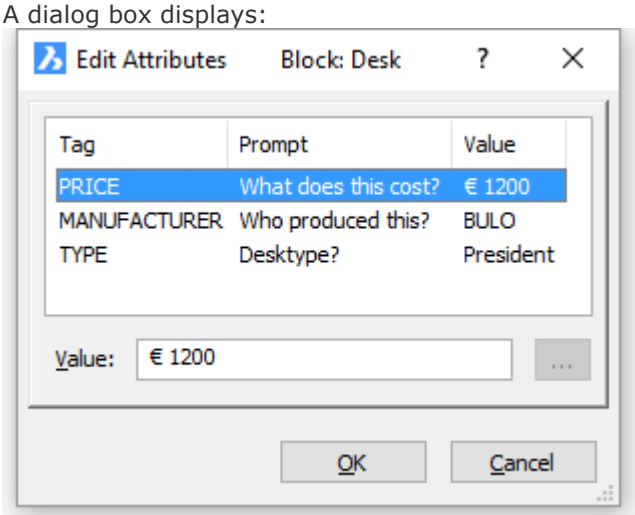


Enter options, and then click OK.

Command Options

Option	Description
Name	Specifies the name of the block. The block can exist in the drawing, or be any DWG file: • To choose a block in the drawing, enter a name, or click the Name droplist. • To choose a DWG file, click the Browse button. Blocks are created with the **Block** command.
Browse	Displays the Open dialog box. Choose a DWG file, and then click Open.

Path	Reports the path to the block, if opened from a DWG file.
Insertion Point	Specifies the insertion point of the block in the drawing. 
Specify On-screen	Toggles where you specify the insertion coordinates: • Off - specifies the x,y,z coordinates in the dialog box. • On - specifies the coordinates in the command bar after you click OK; prompts you: Insertion point - specify the x,y or x,y,z coordinates of the block's lower left corner.
X Y Z	Specifies the x-, y- and z- coordinate of the block's insertion point.
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. 
Specify On-screen	Toggles where you specify the scale factors: • Off - specifies the x,y,z scales in the dialog box. • On - Specifies the scale factor in the command bar after you click OK; prompts you: Scale factor - specify the scale factor.
XYZ	Defines the scaling of the block: • X: Specifies the scale factor along the x axis. • Enter a negative value to mirror the block about the y axis. • Y: Specifies the scale factor along the y axis. • Enter a negative value to mirror the block about the x axis. • Z: Specifies the scale factor along the z axis.
Uniform Scale	Toggles whether or not the same scale factor is used for all: • Yes - uses the x scale factor also for the y and z scale factors. • No - allows the x, y, and z scale factors to be different from each other.
Rotation	Specifies the rotation angle of the block about its insertion point: • Positive angles - rotate the block counter-clockwise.

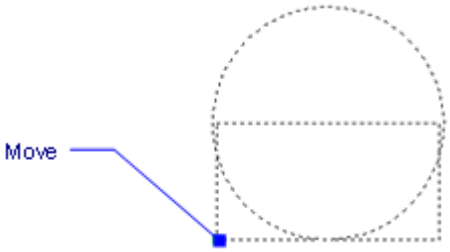
	• Negative angles - rotate the block clockwise. 
Specify On-screen	Toggles where you specify the rotation angle: • Off - Specifies the angle in the dialog box. • On - Specifies the angle in the command bar after you click OK; prompts you: Rotation angle - specify the angle.
Angle	Specifies the rotation angle about the insertion point: • Positive angles rotate the image counter-clockwise. • Negative angles rotate the image clockwise.
Explode	Toggles whether the block is inserted exploded: • Yes - explode block; it is inserted with its constituent parts. • No - does not explode block; it is inserted as a block.
Block Unit	Controls the automatic scaling of the block with respect to the INSUNITS system variable of the current drawing.
Unit	Shows the INSUNITS setting of the inserted block or drawing.
Factor	Shows the calculated scaling factor with respect to the INSUNITS setting of the inserted block (or drawing) and the current drawing.
If the block contains attributes, the value of the ATTDIA (Attribute Dialog) system variable determines whether you are prompted in the command bar to set the attributes or through a dialog box.	
ATTDIA = ON	A dialog box displays:  Select an attribute in the list, then define the attribute in the *Value* field.
ATTDIA = OFF	You are prompted in the command bar: <Attribute Prompt> <Default Text>:

	Type a new value at the command prompt or press Enter to accept the default value.

Grips Editing

Blocks can be edited directly through grips:

1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.



Procedures

General procedure to insert a block
To insert a block
To insert a block using the Drawing Explorer
To insert a block aligned with an entity
Placing blocks with attributes

Related Commands

-Insert - insert blocks and prompts for attribute values through the command bar or a dialog box.
InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
MInsert - inserts a block as a rectangular array
PasteBlock - pastes entities from the Clipboard as blocks.
WBlock - writes blocks and other drawing parts to files on disk.
XAttach - attaches externally-referenced drawings through a dialog box.
AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces blocks to constituent entities and attributes to tags.
Pdfattach - inserts a PDF file as an underlay into the current drawing; displays dialog boxes.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Properties - edits block and attribute values and properties.
RefEdit - edits blocks and attributes in-place.
ResetBlock - resets dynamic blocks to default values.

-Insert

Inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.

Accessing the Command

command bar: *-insert*
alias: *-i*

: ***-insert***
Prompts you in the command bar:

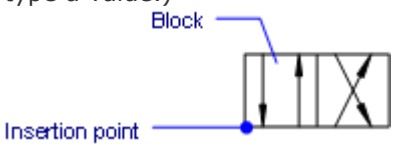
? to list blocks in drawing/~ to open the file dialog/<Block to insert> <BlockName>: (Type the name of the block to insert, press Enter to accept the name of the previously inserted block or choose an option.)

Insertion point for block or [Multiple blocks/Scale]: (Specify the insertion point or choose an option.)

[Corner] <1.000000>: (Specify a point or type a value to specify the X scale factor.)

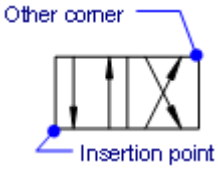
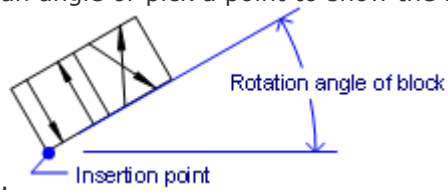
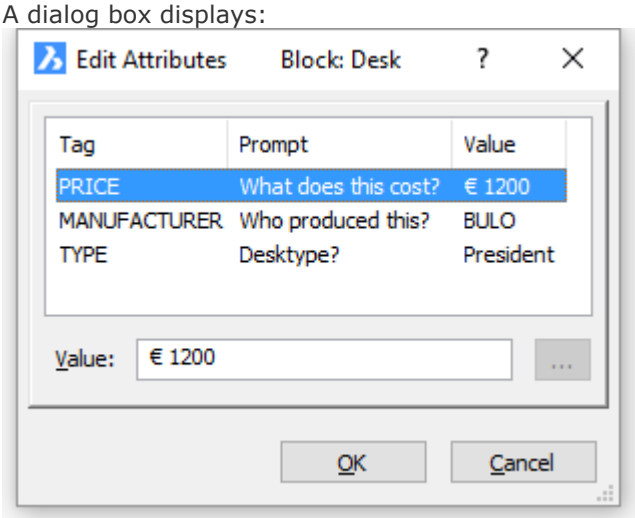
Y scale factor: < Equal to X scale (1.000000)>: (Press Enter to set the Y scale factor equal to the X scale factor or type a value to specify the Y scale factor.)

Rotation angle for block <0>: (Press Enter to accept the default 0° rotation angle, pick a point or type a value.)



Command Options

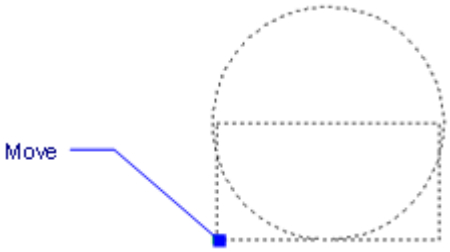
Option	Description
? to list blocks in drawing	Lists the names of all blocks in the current drawing. Prompts you: Block(s) to list <* for all> - press Enter to list all blocks, or enter part of a name to view some blocks. If necessary, press F2 to view the full list. See **TextScr** command.
~ to open the file dialog	Displays the Insert Block dialog box. Choose a .dwg file, and then click Open.
Block to insert	Specifies the name of the block to be inserted. BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the SRCHPATH system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'
Multiple blocks	Repeats the following prompts until you press Enter or Esc. This allows you to insert the same block many times quickly.
Scale	Sets the overall scale factor for the block. Prompts you: Scale factor for block: 1 - Type a value
Insertion point for block	Specifies the insertion point of the block in the drawing.

Corner	Indicates the size of the block by specifying a second corner of a rectangle. The insertion point is the first corner. Prompts you: Select other corner for scale or enter X scale factor - pick a second corner, or enter a number for the x scale factor. You can pick points or enter x,y coordinates. 
Y scale factor	Specifies the scale factor of the block in the y direction. Enter a negative number to mirror the block. Press Enter to make the y scale = x scale.
Rotation angle for block	Specifies the rotation angle of the block about its insertion point. Enter an angle or pick a point to show the rotation 
If the block contains attributes, the value of the ATTDIA (Attribute Dialog) system variable determines whether you are prompted in the command bar to set the attributes or through a dialog box.	
ATTDIA = ON	A dialog box displays:  Select an attribute in the list, then define the attribute in the *Value* field.
ATTDIA = OFF	You are prompted in the command bar: <Attribute Prompt> <Default Text>: Type a new value at the command prompt or press Enter to accept the default value.

Grips Editing

Blocks can be edited directly through grips:

1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.



Related Commands

InsertAligned - inserts blocks repeatedly, and inserts mirrored blocks.
-Insert - insert blocks and prompts for attribute values through the command bar.
PasteBlock - pastes entities from the Clipboard as blocks.
AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces blocks to constituent entities and attributes to tags.
Pdfattach - inserts a PDF file as an underlay into the current drawing; displays dialog boxes.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Properties - edits block and attribute values and properties.
RefEdit - edits blocks and attributes in-place.
ResetBlock - resets dynamic blocks to default values.

InsertAligned

Inserts blocks repeatedly, and inserts mirrored blocks.

Accessing the Command

command bar: *insertaligned*
menu: *Insert | Insert Aligned Block*
alias: *insal*

: ***insertaligned***
Prompts you in the command bar:

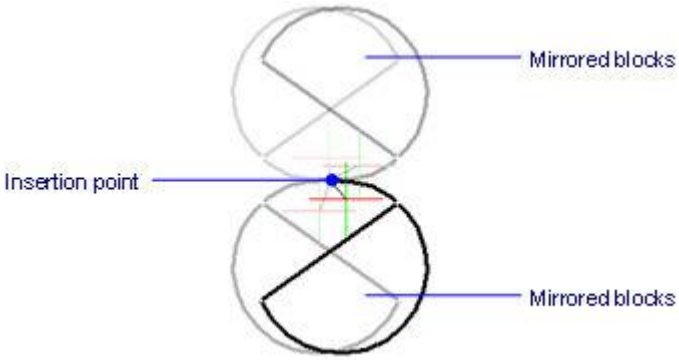
? to list blocks in drawing/~ to open the file dialog/<Block to insert> <defaultname>: (Type ? or ", or enter the name of a block.)

Insertion point for block or [Multiple blocks/Scale]: (pick a point to insert the block once, or type M to insert the block repeatedly.)

Enter block mirroring control point: (Pick a point to mirror the block.)

Command Options

Option	Description
? to list blocks in drawing	Lists the names of all blocks in the current drawing. Prompts you: Block(s) to list <* for all> - press Enter to list all blocks, or enter part of a name to view some blocks. If necessary, press F2 to view the full list. See TextScr command.
~ to open the file dialog	Displays the Insert Block dialog box. Choose a .dwg file, and then click Open.
Block to insert	Specifies the name of the block to be inserted. Press Enter to accept the name of the previously inserted block or type the name of the block to insert.
Multiple blocks	Repeats the Insertion point for next block: prompt until you press Enter or Esc. This allows you to insert the same block many times quickly.
Scale	Allows to scale the block; prompts you: Scale factor for block - type a value.
Insertion point for block	Specifies the insertion point of the block in the drawing.
Enter block mirroring control point	Mirrors the block in one of four quadrants. Move the cursor to locate the mirrored block; the block is mirrored about its insertion point.

Related Commands

Insert - inserts blocks into drawings.
-Insert - insert blocks and prompts for attribute values through the command bar.
AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces blocks to constituent entities and attributes to tags.
Properties - edits block and attribute values and properties.
RefEdit - edits blocks and attributes in-place.

Insertion

Toggles Insertion entity snap; snaps to the insertion point of text and blocks.

Accessing the Command

command bar: *insertion*
transparent: *ins*
toolbar: *Entity Snaps* |

: ***insertion***
Displays no prompts in the command bar.

When insertion snap is on, the cursor snaps to the insertion point of text and blocks.

Command Options

Option	Description
Insertion	Entering this command toggles Insertion entity snap. It snaps to the insertion points of blocks and text. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

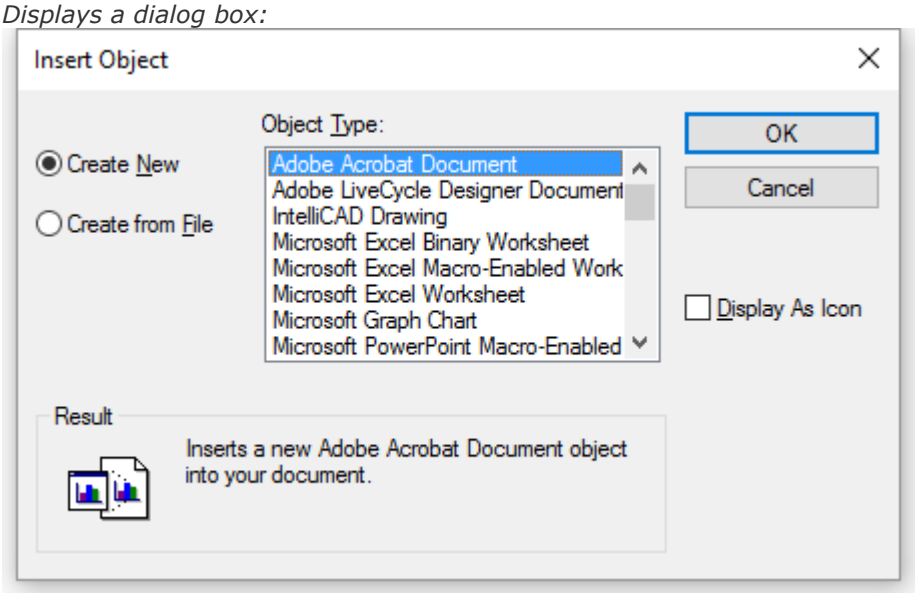
InsertObj

Inserts foreign (non-BricsCAD) objects into drawings.

Accessing the Command

command bar: *insertobj*
alias: *io*
menu bar: *Insert | Insert Object*

: ***insertobj***
Displays a dialog box:



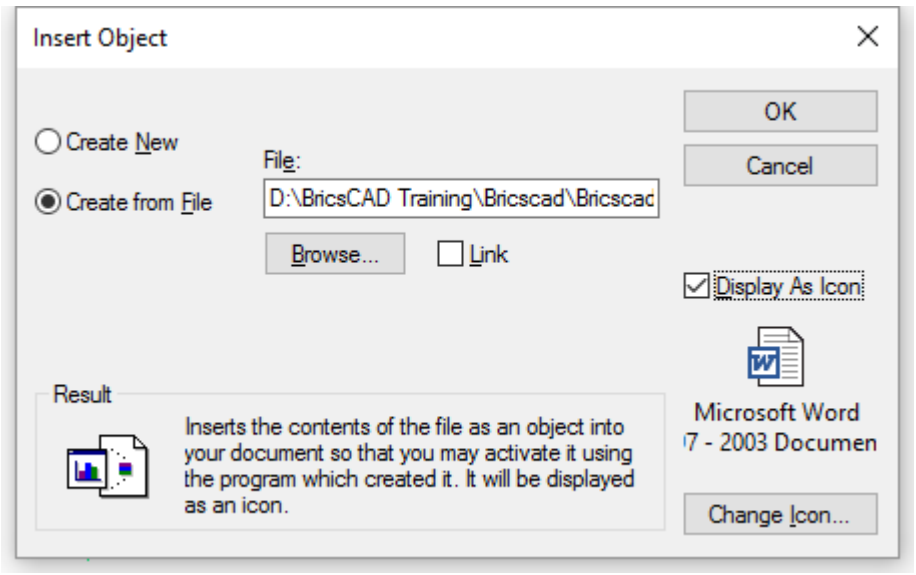
Choose an object type, and then click OK.

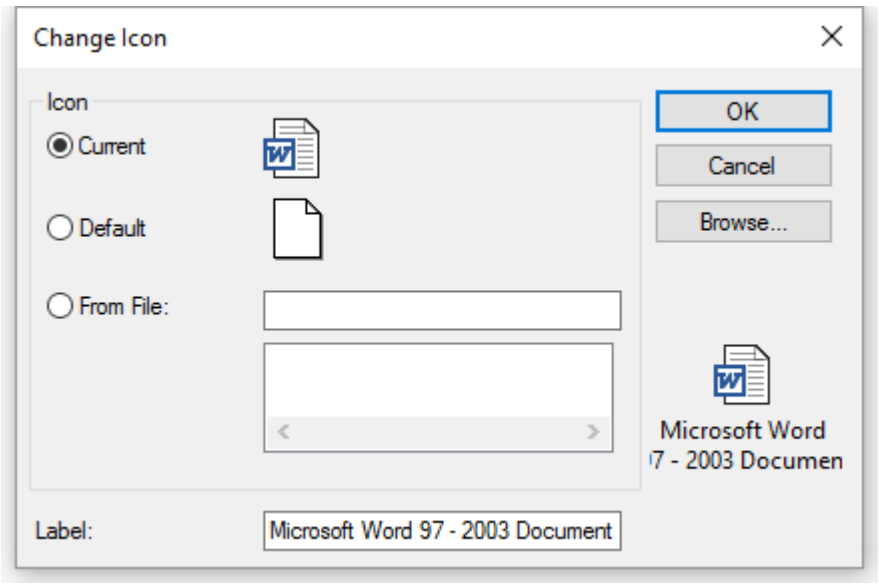
BricsCAD opens the related application; create an object, such as a spreadsheet or diagram.

When done, choose File | Close and Return to Drawing. The object appears in the upper left corner of the drawing.

Command Options

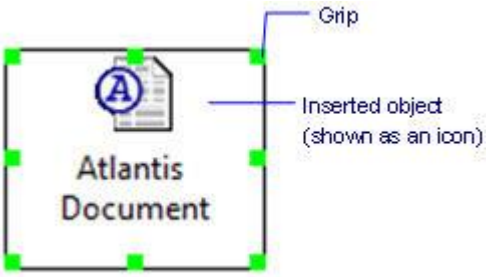
Option	Description
Create New	Creates a new object by opening an application. These objects are also known as OLE objects, short for Object Linking and Embedding.
Create From File	Displays a different set of options:



File	Specifies the file name of the object.
Browse	Displays the Browse dialog box; choose a file, and then click Open.
Link	Toggles linkage to the source document: • On - links the object in the drawing to the source file; when the source file is changed, the linked object also changes. • Off - does not the object.
Display As Icon	Displays the object as an icon. The icon is related to the document type.
Change Icon	Displays the Change Icon dialog box:  Choose another icon, edit the label text, and then click OK.

Grips Editing

Select an OLE object by clicking on its frame. Eight grips appear:



Grips Editing	Description
Resize	Resizes the object by dragging any of the four corner grips. The aspect ratio is maintained.
Stretch	Stretches the object by dragging any of the four midpoint grips. The aspect ratio is not maintained.
Erase	Erases the selected object when the Del button is pressed.

Related Commands

OleLinks - adjusts links of OLE entities embedded in or linked to drawings.
OleOpen - opens linked and embedded objects in the appropriate application.
PasteSpec - pastes entities in several formats.
Insert - inserts DWG files in drawings.
ImageAttach - attaches raster images to drawings.
XAttach - attaches DWG files to drawings.

Interfere

Pro: Shows volumes and areas of interference between ACIS entities.
Platinum: Shows volumes of interference between Acis solids.
Optionally creates new Acis Solids from the common portions of pairs of intersecting solids.

Accessing the Command

command bar: *interfere*
alias: *inf*
menu bar: *Model | 3D Solids| Interfere* and Model | *X-Solids | Tools| Interfere*
toolbar: *3D Solids |*

: ***interfere***
Prompts you in the command bar:

Select first set of Acis objects or [Nested selection] <Nested selection>: (Choose one or more 3D solids or 2D regions.)

Entities in set: 1

Select first set of Acis objects or [Nested selection] <Nested selection>: (Choose more ACIS entities, or press Enter to continue.)

Select second set of Acis objects or [Nested selection] <Nested selection>: (Choose one or more 3D solids or 2D regions.)

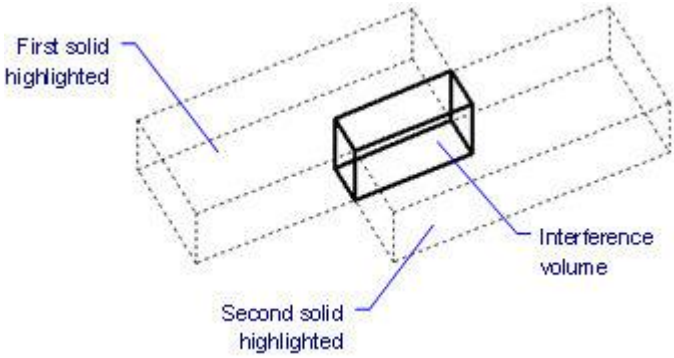
Entities in set: 1

Select second set of Acis objects or [Nested selection] <Nested selection>: (Choose more ACIS entities, or press Enter to continue.)

Create interference solids? Yes/<No>: (Type Y or N.)

Interfering pairs: xx

Highlight pairs of interfering solids? Yes/<No>: (Type Y or N.)



The interference solid remains in the drawing when the command ends.

Command Options

Option	Description
Select first set of Acis objects	Specifies the first set of ACIS entities. Choose one or more 3D solids and/or 2D regions.
Select second set	Specifies the second set of ACIS entities. Choose one or more 3D solids and/or 2D regions.

of Acis objects	The entities in the first set are compared with the entities in the second set.
Nested selection	Allows to selected ACIS entities inside blocks or Xrefs; Prompts you: Select nested Acis objects for first set [eXit] <eXit> - select a nested ACIS entity. Entities in set: 1 Select nested Acis objects for first set [eXit] <eXit> - select mored nested ACIS entities or press Enter to proceed.
Create interference solids?	Determines whether a solid or region of the interference is created: • Yes - creates a solid of the interference volume or region of the interference area. • No - does not create solids or regions of the interference. The solid or region remains after the command ends.
Highlight pairs of interfering solids?	Allows to highlight each pair of intefering ACIS entities; prompts you: Highlight pairs of interfering solids? Yes/<No> - choose Yes to start highlight pairs or No to conclude the command. Enter an option [Next pair/eXit] <Next pair> - choose an option: • Next pair - highlights the next pair. • Exit - exits the command.

Related Commands

Extrude - extrudes 2D entities into 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Region - creates 2D regions from 2D entities.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.

Intersect

Performs Boolean intersection operations on 3D solids and 2D regions; removes all but common portions.

Accessing the Command

command bar: *intersect*
alias: *in*
menu bar: *Model | 3D Solid Editing | Intersect*
toolbar: *3D Solid Editing |*

: ***intersect***
Prompts you in the command bar:

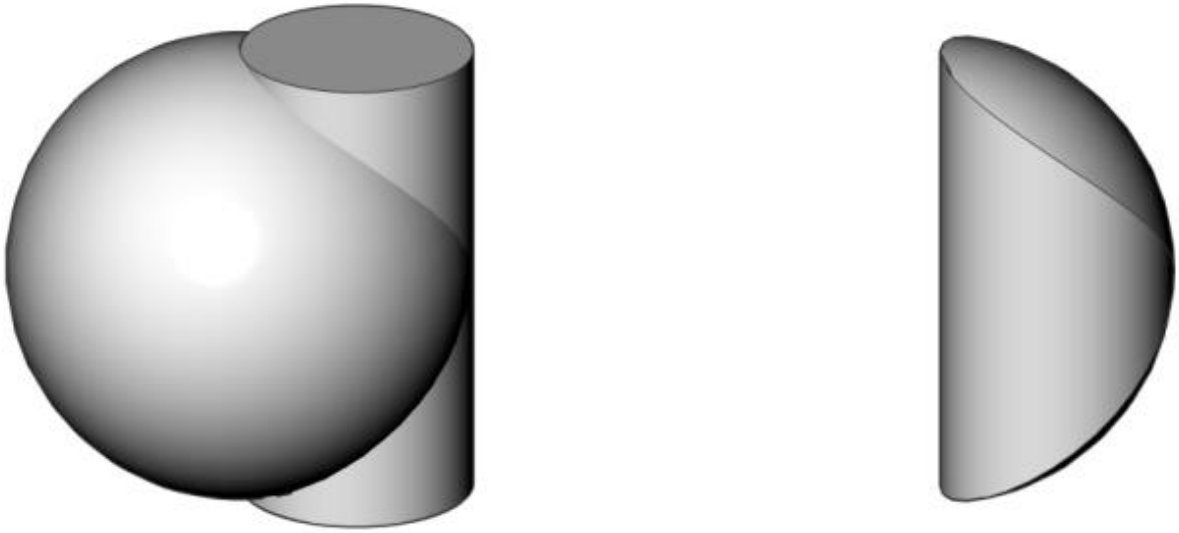
Select ACIS object to intersect: (Choose one 3D solid and/or 2D region.)

Entities in set: 1

Select ACIS object to intersect: (Choose another 3D solid and/or 2D region.)

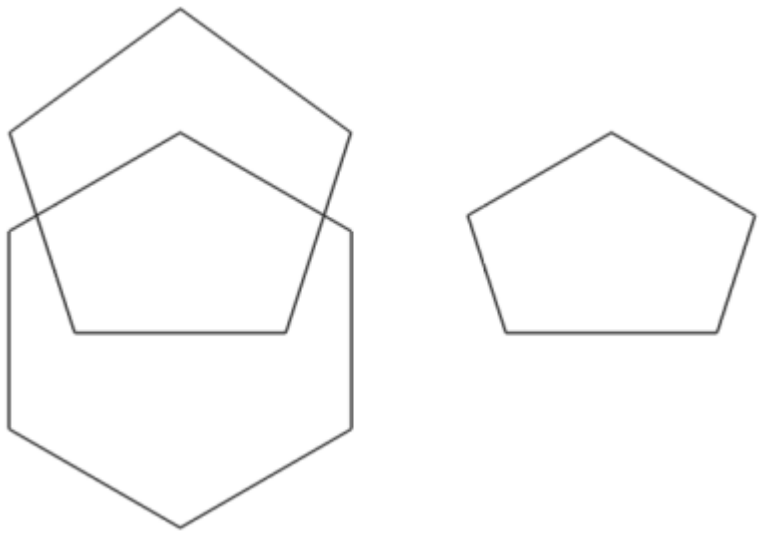
Entities in set: 2

Select ACIS object to intersect: (Choose more ACIS entities, or press Enter to perform the intersection.)



BricsCAD removes all but common volumes and areas from the selected entities: left = selected entities; right = result.

This command also applies to regions:



Caution: if solids and regions do not intersect, BricsCAD erases them.

Command Options

Option	Description
Select ACIS object to intersect	Specifies the 3D solids and/or 2D regions. Choose one or more entities; non-ACIS entities are ignored.

Related Commands

Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Region - creates 2D regions from 2D entities.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.

Intersection

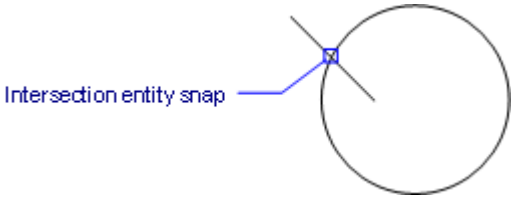
Toggles Intersection entity snap; snaps to the intersections of entities.

Accessing the Command

command bar: *intersection*
transparent: *int*
alias: *3dintersection*
toolbar: *Entity Snaps* |

: ***intersection***

Displays no prompts in the command bar.



When intersection entity snap is on, the cursor snaps to the intersections of crossing entities; it does not snap to apparent intersections in 3D space.

Command Options

Option	Description
Intersection	Entering this command toggles Intersection entity snap. It snaps to the intersections of entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

IsolateObjects

Hides all but the selected entities.

Accessing the Command

command bar: *isolateobjects*
aliases: *isolate*
toolbar: *Hide/Show Entities* |
Quad: *Custom* |

: ***isolateobjects***
Prompts you in the command bar:

Select entities to isolate: : (Select one or more entities.)

Entities in set: 1

Select entities to isolate:: (Select more entities or press Enter.)

Selected entities display only. Other entities are hidden.
The OBJECTISOLATIONMODE system variable controls whether the hidden state of the non-selected entities is saved or not:

[0] Objects are temporarily hidden for the current session. Interfered solids are selected together with the interference solids.
[1] Objects remain hidden between sessions. Interfered solids are selected together with the interference solids.
[2] Objects are temporarily hidden for the current session. Interfered solids are not selected together with the interference solids.
[3] Objects remain hidden between sessions. Interfered solids are not selected together with the interference solids.

Command Options

No options.

Related Commands

LayIso - isolates layers associated with entities selected in the drawing; locks or turns off all other layers.
HideObjects - hides the selected entities.
UnisolateObjects - unhides entities that were hidden.

Isoplane

Switches the isometric drafting plane and grid from left to top to right. Used if SNAPSTYL = ON.

Accessing the Command

command bar: *isoplane*
transparent: *'isoplane*
shortcut: *Ctrl+E, F5*
alias: *is*

: ***isoplane***
Prompts you in the command bar:

Isoplane (Left) [TOGgle/keep using Left/switch to Right/switch to Top] <Switch to Top>: *(Type L, R, or T, or press Enter.)*

Turn on isometric drafting by setting the Snap command's Style option to Isometric.

Turn on the isometric grid with the Grid command's On option.

Draw isometric circles with the Ellipse command's Isocircle option.

Command Options

Option	Description
Left	Switches the isoplane to left.
Right	Switches the isoplane to right.
Top	Switches the isoplane to top.
Toggle	Toggles the isoplane from left to top to right.
Keep using ...	Keep using the current isoplane orientation.

Related Commands

Snap - toggles isometric drawing mode.
Ellipse - draws isometric circles.

J

Join

Joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
The type of the resulting entity depends on the types of the input entities and on their coplanarity.

Accessing the Command

command bar: *join*
menu bar: *Modify | Join*
toolbar: *Modify |*

: ***join***

Prompts you in the command bar:

Select source entity or multiple entities to join at once: (Select one or more entities.)

Select entities to join: (Select more entities or press Enter or right click to join the selected entities.)
BricsCAD reports the result: e.g. 10 entities joined into 2 splines

Command Options

The following rules apply when joining entities:

- Collinear lines will be joined into a single line, even if there are gaps between them.
- Coplanar circular arcs sharing the same radius and center points, will be joined into a circular arc or circle, even if there are gaps between them.
- Coplanar elliptical arcs sharing the same major and minor axes, will be joined into an elliptical arc or ellipse, even if there are gaps between them.
- Circular and elliptical arcs are joined in a counterclockwise direction starting from the source arc.

Examples:

- Two 2D polylines are joined into:
  - a 2D polyline, if they are coplanar.
  - a 3D polyline, if they are not coplanar and if they are composed of straight segments only.
  - a spline, if they are not coplanar and if at least one of the polylines has bulged segments.
- A 2D polyline and a 3D polyline are joined into:
  - a 2D polyline, if they are coplanar.
  - a 3D polyline, if they are not coplanar and if the 2D polyline only has straight segments.
  - a spline, if they are not coplanar and if the 2D polyline has bulged segments.
- A line and a circular arc are joined into,
  - a 2D polyline, if they are coplanar.
  - a spline, if they are not coplanar.
- The following pairs are joined into a spline:
  - a line and an elliptical arc.
  - a spline and an elliptical arc.
  - a helix and a circular arc.

Procedures

To join colinear lines

To join two arcs

Related Commands

Break - breaks entities.
Explode - reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs.
Overkill - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones.
PEdit - joins entities into a polyline.

K

KeepMe

Allows to merge the compared drawings visually.
The command is available during a comparison session only.

Accessing the Command

command bar: *keepme*
toolbar: *Compare* |

: ***keepme***
Displays no prompts you in the command bar:

Command Options

No command options.

Procedures

Merging the compared drawings

Related Commands

DwgCompare - compares a selected drawing with the current drawing during a comparison session.
EndCompare - exits a drawing comparison setting, started by the DwgCompare command.

L

Layer

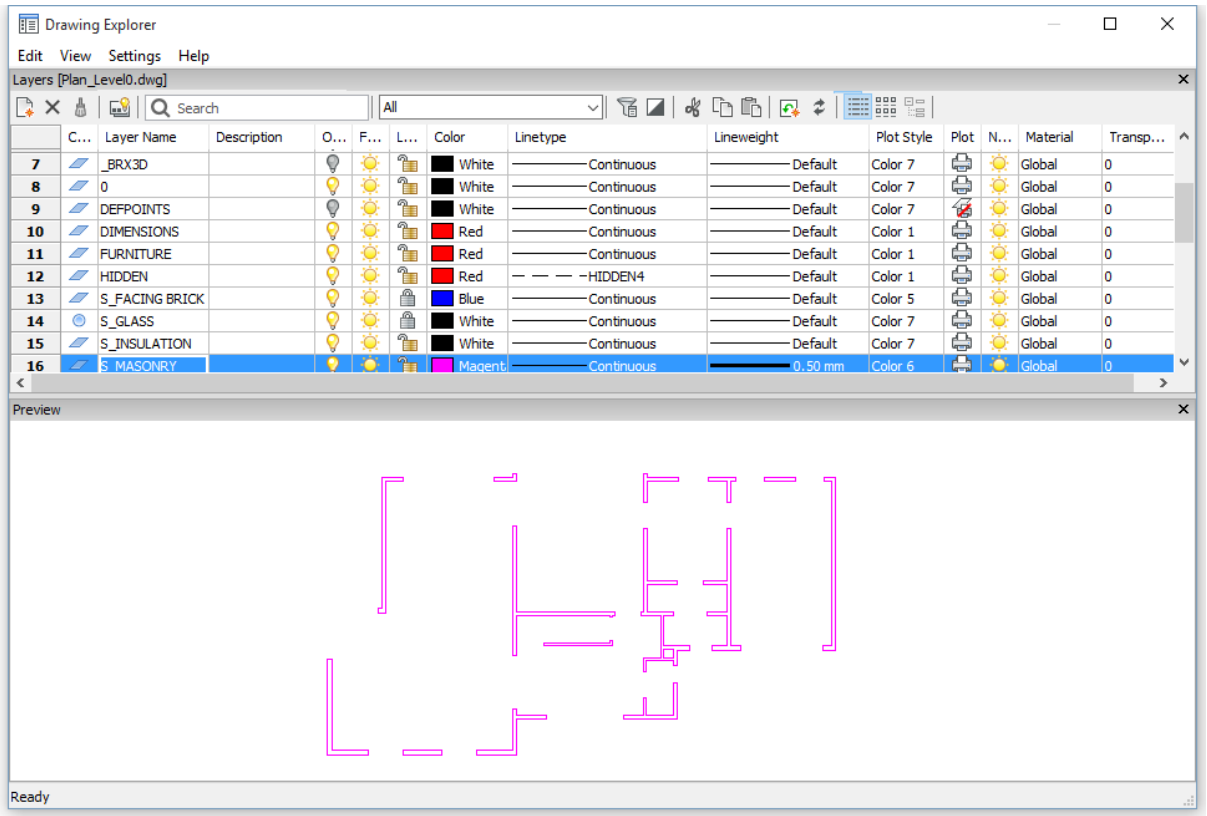
eates layers, changes their properties, and removes layers through the Drawing Explorer.

Accessing the Command

command bar: *layer*
aliases: *la, ddlmodes, explayers*
menu bar: *Settings | Layers* and *Tools | Drawing Explorer | Layers*
toolbar: *Entity Properties |* and *Layers |*
status bar: *Layer* field (*0*)

: ***layer***

Displays the Layers section of the Drawing Explorer dialog box:



Create layers or modify layer properties, and then click ***X.***

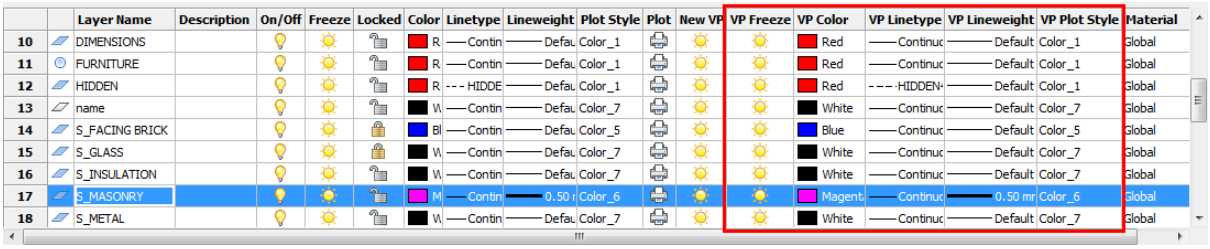
*Press **F1** to access help.*

Viewport Layer Properties

When you open the *Drawing Explorer - Layers* dialog when in paper space, additional columns display: *New*, *VP Freeze*, *VP Color*, *VP Linetype*, *VP Lineweight* and *VP Plot Style* (VP is short for Viewport). By default, these columns match the corresponding model space settings. *VP* layer properties can be set differently for each layout and for each paper space viewport.

If SHOWLAYERUSAGE is on, the *Layer Usage* icons indicate whether *VP* settings for the current layout or paper space viewport are different from the model space settings:

- : Current layer with VP overrides.
- : Layer with VP overrides.
- : Empty layer with VP overrides.



	Layer Name	Description	On/Off	Freeze	Locked	Color	Linetype	Lineweight	Plot Style	Plot	New VP	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Plot Style	Material
10	DIMENSIONS					R	Contin	Defau	Color_1				Red	Continuc	Default	Color_1	Global
11	FURNITURE					R	Contin	Defau	Color_1				Red	Continuc	Default	Color_1	Global
12	HIDDEN					R	HIDDE	Defau	Color_1				Red	HIDDEN	Default	Color_1	Global
13	name					W	Contin	Defau	Color_7				White	Continuc	Default	Color_7	Global
14	S_FACING BRICK					Bl	Contin	Defau	Color_5				Blue	Continuc	Default	Color_5	Global
15	S_GLASS					W	Contin	Defau	Color_7				White	Continuc	Default	Color_7	Global
16	S_INSULATION					W	Contin	Defau	Color_7				White	Continuc	Default	Color_7	Global
17	S_MASONRY					M	Contin	0.50 r	Color_6				Magent	Continuc	0.50 mr	Color_6	Global
18	S_METAL					W	Contin	Defau	Color_7				White	Continuc	Default	Color_7	Global

- If no paper space viewport is active, the *VP* settings apply to entities in the current layout.
- If a viewport is active, the *VP* settings override the *BYLAYER* properties in the current viewport.
- The *VP Freeze* setting (or)allows to control the visibility of model space entities in the current viewport.
- The New setting controls whether a layer is either Frozen ( ) are Thawed ( ) in a new paper space viewport.

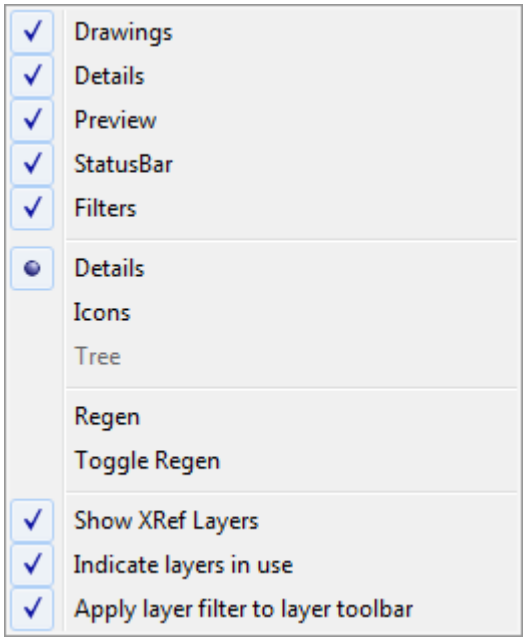
Create or modify VP layer properties, and then click X.

Press F1 to access help.

To remove VP Layer properties overrides:

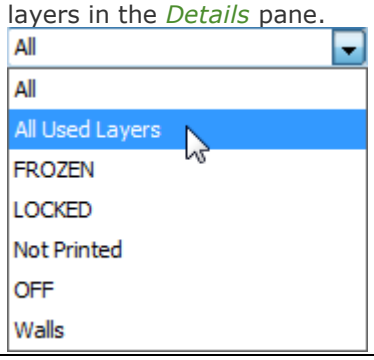
1. Select the layer(s), then right click and choose Remove Viewport Overrides in the context menu.
2. The options are:
   3. From Selected Layers
      - For Current Viewport
      - For All Viewports
   4. From All Layers
      - For Current Viewport
      - For All Viewports

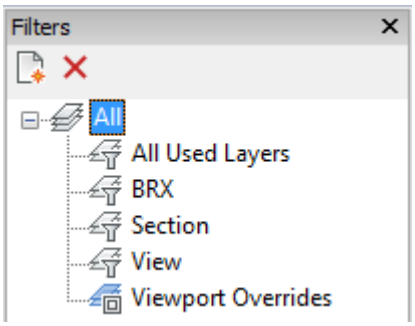
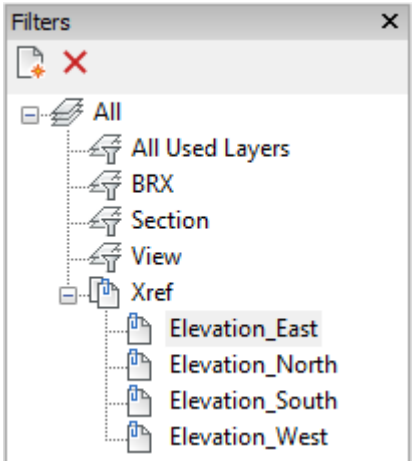
View Menu

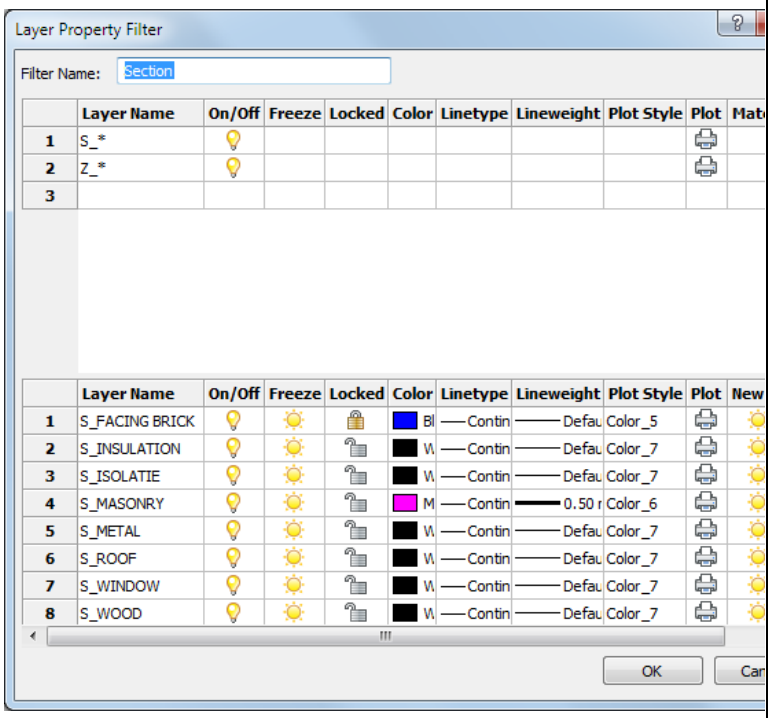


Option	Description
Drawings	Toggles the display of the *Drawings* panel.
Details	Toggles the display of the *Details* panel.
Preview	Toggles the display of the *Preview* panel.
StatusBar	Toggles the display of the *Status Bar*.
Filters	Toggles the display of the *Filters* panel.
Details	Switches the display in the *Details* panel to *Details* view.
Icons	Switches the display in the *Details* panel to *Icons* view.
Regen	Forces the drawing display to be regenerated.
Toggle Regen	Toggles the automatic regen of the drawing display.
Show XRef Layers	Toggles the display of Xref layers in the *Details* panel.
Indicate layers in use	Toggles the SHOWLAYERUSAGE system variable On/Off.
Apply layer filter to layer toolbar	Applies the current layer filter to the *Layer* control on the *Entity Properties* toolbar.

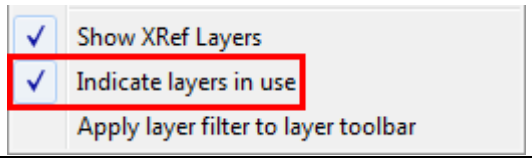
Command Options

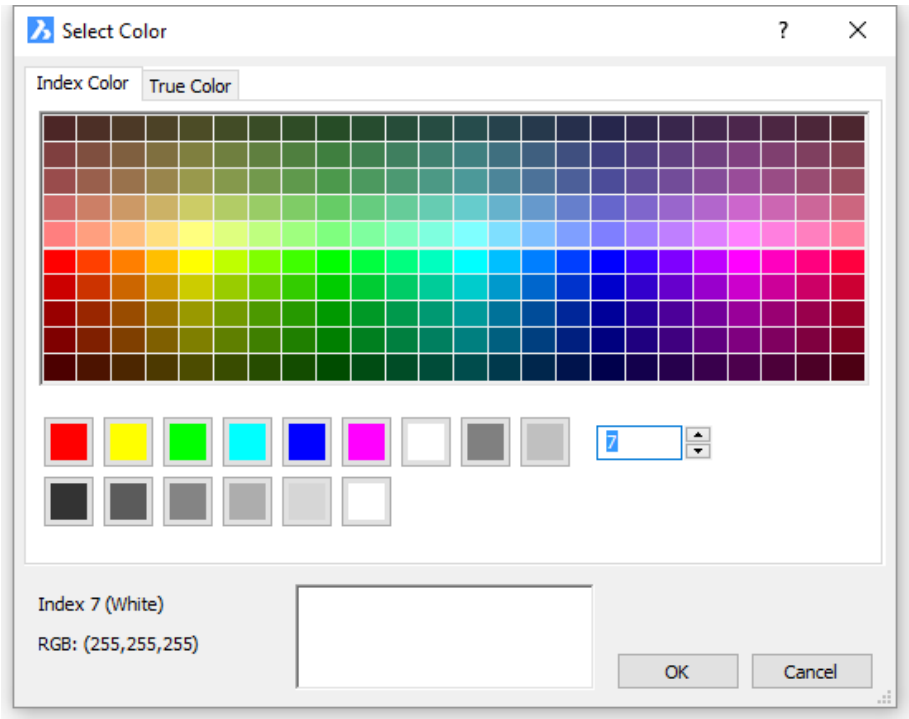
Option	Description
New	Creates new layers. A new layer is created as a copy of the currently selected layer.
Delete	Deletes layers. The following layers cannot be deleted: • 0 • Any layer with entities.
Purge	Purges layers from the current drawing. The following layers cannot be purged: • 0 • Current layer. • Any layer with entities.
Xref symbols On/Off	Allows to toggle the display of Xref layers.
Search	Allows to filter the layer list. Only layernames which contain the text string in the *Search* field are listed.
Filter Used	Displays the current layer filter. Click the down-arrow button to choose a different layer filter. *All* and *All Used Layers* are built-in layer filters. Choose *All* to see all layers in the list; choose *All Used Layers* to list non-empty layers only. If the drawing contains external references (Xrefs) a layer filter definitions for Xref layers and for each Xref are created automatically. Choose Show XRef Layers in the *View* menu to display the Xref layers in the *Details* pane. 

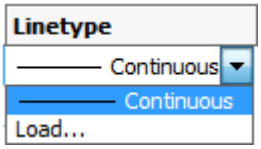
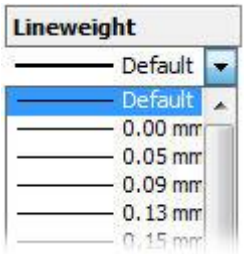
Display Filter Panel	To toggle the display of the *Filter Panel* On/Off do one of the following: • Click the *Display Filter Panel* button ( ). • Check/uncheck the *Filters* option in the *View* menu.  If the drawing contains external references (Xrefs) a layer filter definitions for Xref layers and for each Xref are created automatically. Choose Show XRef Layers in the *View* menu to display the Xref layers in the *Details* pane.  On the *Filter Panel*: • Select a filter in the list to apply the filter. The caption on the *Filter Used* list button displays the selected filter. • Click the *New* button ( ) to create a new layer filter definition. • Click the *Delete* button ( ) to delete the selected filter(s). The *All Used Layers* built-in filter cannot be deleted. To create a new layer filter definition: 1. Click the *New* button ( ) on the *Filter Panel* toolbar. The *Layer Property Filter* dialog box displays:

<table>
<tr>
<td></td>
<td>


2. Type a search string in the Filter Name field.
Accepted wild-card characters are listed in the Wild-Card Table below.
3. (option) Type a search string in the Layer Name field (see Filter Text).
4. (option) Click a property column, then set a filter criterion for this property.
Multiple properties can have criterion applied.
5. (option) Repeat steps 3 and 4 for each new row in the layer filter definition.
6. Click the OK button to save the layer filter.
</td>
</tr>
<tr>
<td>Invert Filter</td>
<td>Click the Invert Filter button to invert the current filter.</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</table>

Current	Specifies the current working layer. Only one layer can be current at a time.
	If the SHOWLAYERUSAGE system variable is ON, the following icons display in the *Layer Usage* column: • layer is current • layer is not empty • layer is empty When in a paper space layout or in a paper space viewport (model space with floating viewports) the following icons indicate VP overrides exist: • VP overrides exist and layer is current • VP overrides exist and layer is not empty • VP overrides exist and layer is empty The *Indicate layers in use* item in the View menu toggles the SHOWLAYERUSAGE system variable ON/Off. 
Layer Name	Specifies the name of the layer. • Click in the column, then type a character to jump to the first layer starting with that character. • To rename: 1. Click the name twice, slowly. 2. Enter the new layer name.
Description	(option) Describes the content of the layer. Click the field, then type.
On/Off	Toggles the selected layers between on and off: On - sets the normal state. Off - hides entities from view; entities are included in the calculation of extents. Click a lightbulb icon to change its status.
Freeze	Toggles the selected layers between frozen and thawed: Thaw - sets the normal state. Freeze - hides entities from view; entities are excluded in the calculation of extents, and are not exported with commands such as WBlock. Click an icon to toggle its status.
Locked	Toggles the selected layers between locked and unlocked: Unlocked - sets the normal state. Locked - prevents entities from being edited; entities are still visible.

	Click an icon to toggle its status.
Color	Specifies the color of the layer. Default color is white (black). To change the color: 1. Click a color square to see the Select Color dialog box.  2. Choose a color, and then click OK. See **Color** command.

Linetype	Specifies the linetype of selected layers. Default is "Continuous." To change the linetype: 1. Click a linetype sample. 2. Choose one from the droplist.  To load additional linetype definitions: 1. Click the Load option. 2. In the Load Linetypes dialog box, choose a linetype, and then click OK. See **Linetype** command.
Lineweight	Specifies the linetype of selected layers. Default is "Default." To change the lineweight: 1. Click the lineweight sample. 2. From the droplist, choose another linetweight.  See **LWeight** command.
Plot Style	Specifies the plot style of selected layers. Default is "Color_7." To change the plot style" 1. Click a plot style name. 2. From the droplist, choose another style name. Plot styles are available only when they are enabled in the drawing; by default, they are not. See **PlotStyle** command.
Plot	Toggles the selected layers between plotable and not plotable: • Plot - plots layer. • No Plot - does not plot layer. Click an icon to toggle its status.
New Viewport	Specifies whether the layer is visible (thawed) or invisible (frozen) in a new layout viewport.
Material	Specifies the material of selected layers. Default is "Global." To change the material: • Click the Material field. • Click again to display the droplist, containing the available materials. • Click to select a material in the list.

Transparency	Sets the Transparency factor between 0 (fully opaque) and 90 (fully transparent). The TRANSPARENCYDISPLAY system variable controls the display of the transparency in the *2DWireframe* visual style. Rendered visual styles always display transparency.
Preview	Displays the entities on the selected layer.

Wild-Card Characters

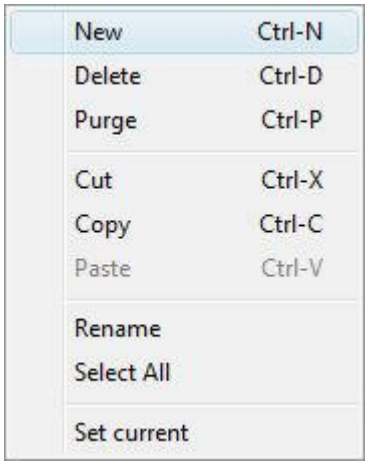
Wild-card characters can be used in the Filter text field and in Filter definitions to filter layers by name.

Character	Name	ASCII code*	Result	Example
#	pound	35	Matches numeric digits.	## matches any 2 digit number: 12 or 34 but not 5 or 123
@	at	64	Matches alphabetic characters.	@@@@@ matches all 5-letter layer names
.	period	46	Matches non-alphanumeric characters.	.* matches all layer names starting with an non-alphanumeric character: _SITEPLAN or +1_LEVEL
*	asterisk	42	Matches any character string.	*A* matches all layer names containing the letter A
?	question mark	63	Matches any single character.	?AB* matches LABORATORY and CABARET but not STABLE
~	tilde	126	Matches any string but the pattern.	~*A* matches all layer names not containing the letter A
[]	brackets	91 93	Matches any one of the characters enclosed.	LA[YB] matches LAY and LAB
[~]			Matches any one of the characters not enclosed.	LA[~YB] matches LAD but not LAY or LAB
[-]			Specifies a range for a single character.	LAY-[A-E] matches any combination of LAY- and A through E: LAY-A ... LAY-E but not LAY-F
`	reverse quote	96	Matches the next string exactly.	Allows to find a layer name that contains a wild-card character. If preceded with a reverse quote, it is not interpreted as a wild-card character. `~LAY* matches ~LAY-1,~LAYER-1, ~LAY-B, &ldots;

* ASCII code: if a character is not available on the keyboard, press and hold the Alt-key and type the ASCII code on the numeric keypad.

Shortcut Menu Options

Right-click layer names in Drawing Explorer to see the following shortcut menu:



Option	Keystroke	Description
New	Ctrl+N	Creates new layers.
Delete	Ctrl+D	Deletes layers from Drawing Explorer. The following layers cannot be deleted: • 0 • Any layer with entities.
Purge	Ctrl+P	Purges layers from the current drawing. The following layers cannot be purged: • 0 • Current layer. • Any layer with entities.
Cut	Ctrl+X	Cuts the layer and its properties to the Clipboard.
Copy	Ctrl+C	Copies the layer and its properties to the Clipboard.
Paste	Ctrl+V	Pastes the layer from the Clipboard into Drawing Explorer.
Rename		Renames the selected layer.
Select All		Selects all layers.
Set Current		Specifies the current working layer. Only one layer can be current at a time.

Status Bar Menu

Right-click O (or other layer name) on the status bar for the shortcut menu:

Shortcut Menu	Description
0, et cetera	Chooses the current working layer. When the drawing contains additional layers, their names are listed here.
Properties	Opens the Layers section of the Drawing Explorer.

Procedures

To merge the content of two layers

Related Commands

-Layer - creates layers and changes their properties in the command bar.
LayersPanelOpen - opens the Layers panel, which allows to edit the layers in the current drawing, apply layer states and layer filters.
LayersPanelClose - closes the Layers panel.
LayerState - saves and restores layer states.
LayMCur - changes the working layer to that of a selected entity
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.
Materials - creates materials and edits their properties through the Drawing Explorer.
VpLayer - changes the properties of layers in the current paperspace viewport.

Explorer - Opens the Drawing Explorer dialog box.

-Layer

Creates layers, changes their properties, and removes layers through prompts in the command bar.

Accessing the Command

command bar: *-layer*
alias: *-la*
status bar: *0*

: ***-layer***
Prompts you in the command bar:

Layer [? to list/New layer/Make new current layer/Set layer as current/Color/Linetype/LineWeight/TRansparency/MATerial/Plot/stAte/turn layer ON/turn layer OFf/Freeze/Thaw/LOck/Unlock]:

This command repeats itself until you press Enter at the prompt.

Command Options

Option	Description
? to list	Lists the names of layers in the current drawing.
New layer	Creates new layers; prompts you: Names for new layers - enter one or more layer names. To create several new layers at the same time, separate their names with commas, such as: doors,windows,walls
Make new current layer	Creates a new layer and then makes it current; prompts you: New layer to make current - enter a layer name. This option combines the New and Set options.
Set layer as current	Sets a layer as the current one; prompts you: New layer to make current - enter a layer name.
Color	Changes the color of layers; prompts you: Color - specify the color by number or name. See Color command. Layer(s) to be *color* - specify the layer names; separate layer names with commas.
Linetype	Changes the linetype of layers; prompts you: Linetype: ? to list <Continuous> - enter an option. See the Linetype and -Linetype commands. • ? to list - Lists the names of linetypes defined in the current drawing. • <Continuous> - Specifies the name of the linetype to assign to the layer. Layer(s) for linetype CONTINUOUS <CurrentLayer> - Specify the layer names; separate layer names with commas.
Lineweight	Changes the lineweight of layers; prompts you: **Enter lineweight (0.00 - 2.11mm)** - enter a number between 0.00 and 2.11.

	Enter name list of layers(s) for lineweight 0.00 mm <CurrentLayer> - Specify the layer names; separate layer names with commas.

Transparency	Sets the Transparency factor between 0 (fully opaque) and 90 (fully transparent); prompts you: Enter transparency (0 - 90) - enter a number between 0 and 90. Layer(s) for transparency 0 <CurrentLayer> - Specify the layer names; separate layer names with commas. The TRANSPARENCYDISPLAY system variable controls the display of the transparency in the *2DWireframe* visual style. Rendered visual styles always display transparency.
Material	Sets the Material property; prompts you: Material: ? to list/<Global> - enter an option. • ? to list - Displays a list of the materials and their description in the *Prompt History* window. • Global - Sets the *Material* property to the *Global* material, as defined in the Drawing Explorer - Materials. All drawings have a *Global* material, which cannot be deleted or renamed. Layer(s) for material GLOBAL <CurrentLayer> - Specify the layer names; separate layer names with commas.
Plot	Changes the plot status of layers (plot or no plot); prompts you: Enter a plotting preference [Plot/No] <Plot> - enter an option. Enter layer name(s) for this plot preference <CurrentLayer> - Specify the layer names; separate layer names with commas.

State	Allows creating, editing, renaming, restoring, deleting, importing and exporting layer states. See the LayerState command.
	Prompts you:
	Enter an option [? to list layer states/Save/Restore/Edit/reName/Delete/Import/eXport] - Enter an option.
	• ? to list layer states - Displays a list of the saved layer states and their description in the Prompt History window.
	• Save - Saves the current layer state. Prompts you:
	  • Enter a name for the new Layer State - Type a name.
	  • Enter state to change [On/Frozen/Lock/Color/lineType/lineWeight/plot Style/Plot/New vp frozen] - All states are On by default. Enter a state to toggle the current setting.
	  Enter another option or press Enter to create the new layer state.
	• Restore - Restores a saved layer state. Prompts you:
	  • Enter name of Layer State to restore or [?] - Type the name of saved layer state or type ? to list the saved layer states.
	• Edit - Edits a saved layer state. Prompts you:
	  • Enter the name of the Layer State to edit or [?] - Type the name of saved layer state or type ? to list the saved layer states.
	• Rename - Renames a saved layer state. Prompts you:

	• Type the name of saved layer state or type ? to list the saved layer states - Type the name of saved layer state or type ? to list the saved layer states. • Enter the new name - type the new name for the selected layer state. • Delete - Deletes a saved layer state. Prompts you: • Enter the name of the Layer State to delete or [?] - Type the name of saved layer state or type ? to list the saved layer states. • Import - Imports a layer state from a file into the current drawing. Displays a File dialog. Select a layer state file (*.las) then press the *Open* button. • Export - Exports a saved layer state in the current drawing to a layer state file (*.las). Prompts you: • Enter the name of the Layer State to export or [?] - Type the name of saved layer state or type ? to list the saved layer states. • Displays a File dialog - Type a name for the layer state file, then press the *Save* button.
Turn layer ON	Turns layers on; prompts you: Layer(s) to turn ON - Specify the layer names; separate layer names with commas. This option is the compliment to the OFF option.
Turn layer OFF	Turns layers off; prompts you: Layer(s) to turn OFF - Specify the layer names; separate layer names with commas. When layers are off, they are hidden from view. Entities on these layers are taken into account when BricsCAD calculates the extents of the drawing. When you attempt to turn off the current layer, BricsCAD prompts you: LAYER is the CURRENT layer. Turn it off anyway? <N> - enter Y or N.
Freeze	Freezes layers; prompts you: Layer(s) to Freeze - Specify the layer names; separate layer names with commas. When layers are frozen, they are hidden from view and cannot be edited. Entities on these layers are ignored when BricsCAD calculates the drawing extents. The current layer cannot be frozen; BricsCAD warns you: **Layer is the CURRENT layer. It cannot be frozen.**
Thaw	Thaws frozen layers; prompts you: Layer(s) to Thaw - Specify the layer names; separate layer names with commas. This option is the complement to the Freeze option.
Lock	Locks layers; prompts you: Layer(s) to Lock - Specify the layer names; separate layer names with commas. When layers are locked, their entities can be seen but not edited. The current layer can be locked.
Unlock	Unlocks locked layers; prompts you: Layer(s) to Unlock - Specify the layer names; separate layer names with commas.

	This option is the compliment to the Lock option.

Status Bar Menu

Right-click the *Current Layer* field on the status bar for the shortcut menu:

Shortcut Menu	Description
0 et cetera	Chooses the current working layer. When the drawing contains additional layers, their names are listed here.
Properties	Opens the Layers section of the Drawing Explorer. See Layer command.

Related Commands

Layer - creates layers and changes their properties in the Drawing Explorer.
LayersPanelOpen - opens the Layers panel, which allows to edit the layers in the current drawing.
LayersPanelClose - closes the Layers panel.
LayerState - saves and restores layer states.
LayMCur - changes the working layer to that of a selected entity
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayFrz

Freezes the layers associated with entities selected in the drawing (short for "layer freeze").

Accessing the Command

command bar: *layfrz*
menu bar: *Settings | Layers | Freeze Layer*
toolbar: *Layers |*

: ***layfrz***
Prompts you in the command bar:

Current setting: Viewports=Freeze, Block nesting level=Block

Select an entity on the layer to be frozen or [Settings]: (Pick an entity, or type S.)

This layer has been frozen: LAYERNAME.

Select an entity on the layer to be frozen or [Settings]: (Pick another entity, or press Enter to exit the command.)

The current layer cannot be frozen; BricsCAD warns you:
Cannot freeze layer. It is the current layer.

Command Options

Option	Description
Select an entity on the layer to be frozen	Selects an entity. Choose an entity whose layer should be frozen. This prompt repeats itself until you press Enter; this allows you to freeze more than one layer.
Settings	Specifies how viewports, blocks, and xrefs should be frozen; prompts you: Enter setting type for [Viewports/Block selection] - type V or B.
Viewports	Specifies how viewports should be frozen; prompts you: In paper space viewport use [Freeze/Vpfreeze] - type F or V. • Freeze - freezes layers in all viewports (default). • Vpfreeze - freezes layers in the current viewport only.
Block selection	Specifies how blocks and xrefs should be frozen; prompts you: Enter Block Selection nesting level [Block/Entity/None] - type B, E, or N. • Block - freezes layers associated with the block insertion or xref attachment (default). • Entity - freezes layers associated with entities selected in the block or xref. (Entities that make up blocks can sometimes be on different layers.) • None - does not freeze the block or xref.

Related Commands

Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayThw

Thaws all frozen layers (short for "layer thaw").

Accessing the Command

command bar: *laythw*
menu bar: *Settings | Layers | Thaw All Layers*
toolbar: *Layers |*

: ***laythw***
BricsCAD responds in the command bar:

All layers have been thawed.

Layers which are turned Off are not affected.

Command Options

No options.

Related Commands

LayFrz - freezes layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayIso

Isolates layers associated with entities selected in the drawing; locks or turns off all other layers (short for "layer isolate").

Accessing the Command

command bar: *layiso*
menu bar: *Settings | Layers | Isolate Layer*
toolbar: *Layers |*

: ***layiso***

Prompts you in the command bar:

Current setting: Lock layers

Select entities on the layer(s) to be isolated or [**Settings**]: (Pick one or more entities, or type S.)

Number of found objects: 1, number of selected objects: 1

Select entities on the layer(s) to be isolated or [Settings]: (Choose more entities, or press Enter to exit the command.)

Number of isolated layers: xx.

If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed.
BricsCAD reports in the command bar:
Number of isolated layers: xx. This layer is current: LAYER_NAME.

The LAYUNISO command restores the layer state that was current before the LAYISO command was launched.

Command Options

Option	Description
Select entities on the layer(s) to be isolated	Selects one or more entities. Choose the entities whose layers should be not be locked or turned off. This prompt repeats itself until you press Enter; this allows you to isolate more than one layer.
Settings	Specifies how layers and viewports should be isolated; prompts you: Enter setting for layers not isolated [Off/Lock] - type O or L. • Off - all other layers are turned off (made invisible). • Lock - all other layers are locked (visible but cannot be edited). In paper space viewport use [Vpfreeze/Off] - type V or O. • Vpfreeze - freezes layers in the current viewport. • Off - turns off layers.

Related Commands

Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayLck

Locks the layer associated with an entity selected in the drawing; locked layers can be seen but not edited (short for "layer lock").

Accessing the Command

command bar: *laylck*
menu bar: *Settings | Layers | Lock Layer*
toolbar: *Layers |*

: ***laylck***
Prompts you in the command bar:

Select an entity on the layer to be locked: **(Choose an entity.)**
This layer has been locked: <LayerName>

When you choose an entity on an locked layer, BricsCAD reports:

This layer is already locked.

The LAYERLOCKFADECTL system variable sets the fading of locked layers. Values between 0 (no fading) and 90 are accepted.

Command Options

No Options.

Related Commands

LayUlk - unlocks layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayUlk

Unlocks the layer associated with an entity selected in the drawing (short for "layer unlock").

Accessing the Command

command bar: *layulk*
menu bar: *Settings | Layers | Unlock Layer*
toolbar: *Layers |*

: ***layulk***
Prompts you in the command bar:

Select an entity on the layer to be unlocked: *(Choose an entity.)*
This layer has been unlocked: <LayerName>

When you choose an entity on an unlocked layer, BricsCAD reports:
This layer is already unlocked.

Command Options

Option	Description
Select an entity on the layer to be unlocked	Selects an entity. Choose an entity whose layer should be unlocked.

Related Commands

LayLck - locks layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayOff

Turns off layers associated with entities selected in the drawing; off layers cannot be seen (short for "layer off").

Accessing the Command

command bar: *layoff*
menu bar: *Settings | Layers | Layer Off*
toolbar: *Layers |*

: ***layoff***
Prompts you in the command bar:

Current setting: Viewports=Off, Block nesting level=Block

Select an entity on the layer to be turned off or [Settings]: *(Choose an entity, or type S.)*
This layer has been turned off: <LayerName>.
Select an entity on the layer to be turned off or [Settings]: *(Choose another entity, or press Enter to exit the command.)*

Command Options

Option	Description
Select an entity on the layer to be turned off	Selects an entity. Choose an entity whose layer should be turned off. This prompt repeats itself until you press Enter; this allows you to turn off more than one layer.
Settings	Specifies how viewports, blocks, and xrefs should be turned off; prompts you: **Enter setting type for [Viewports/Block selection]** - type V or B.
Viewports	Specifies how layers and viewports should be turned off; prompts you: **In paper space viewports use [Vpfreeze/Off]** - type V or O. • **Vpfreeze** - freezes the layers in the current viewport. • **Off** - turns off layers (made invisible).
Block selection	Specifies how blocks and xrefs should be frozen; prompts you: **Enter Block Selection nesting level [Block/Entity/None]** - type B, E, or N. • **Block** - freezes layers associated with the block insertion or xref attachment (default). • **Entity** - freezes layers associated with entities selected in the block or xref. (Entities that make up blocks can sometimes be on different layers.) • **None** - does not freeze the block or xref.

Related Commands

LayOn - turns on all layers that have been turned off.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.

LayOn

Turns on all off layers (short for "layer off").

Accessing the Command

command bar: *layon*
menu bar: *Settings | Layers | Layer On*
toolbar: *Layers |*

: ***layon***
BricsCAD responds in the command bar:
All layers have been turned on.

This command turns ON all layers, whether turned off by the LayOff or by other commands. Frozen layers are not affected.

Command Options

No options.

Related Commands

LayOff - turns off layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.

LayUnIso

Restores layers isolated by the LayIso command (short for "layer unisolate").

Accessing the Command

command bar: *layuniso*
menu bar: *Settings | Layers | Unisolate Layer*
toolbar: *Layers |*

: ***layuniso***
BricsCAD responds in the command bar:
Layers isolated by LAYISO command have been restored.

Command Options

No options.

Related Commands

LayOff - turns off layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.

LayCur

Moves the selected entities to the current layer.

Accessing the Command

command bar: *laycur*
toolbar: *Entity Properties* |

: ***laycur***
Prompts you in the command bar:

Select entities to be moved to the current layer: (Select entities)

Entities in set: xx

Select entities to be moved to the current layer: (Select more entities or right click to move the selection set to the current layer)

xx entities were moved to the current layer ("<Layer Name>")

Command Options

No options.

Related Commands

Change - changes the properties of entities, through the command bar.
ChProp - changes the properties of entities, with fewer options than the Change command

LayerP

Undoes previously applied changes to layer settings if LAYERPMODE is ON. Short for Layer Previous.

Accessing the Command

command bar: *layerp*

: ***layerp***

Previous layer settings were restored.

The following changes cannot be undone by the LayerP command:

- Changes made to layer names.
- Deleted or purged layers are not restored.
- Added layers: new layers are not removed.

Command Options

No options.

Related Commands

Layer - creates layers, changes their properties, and removes layers through the Drawing Explorer.
-Layer - creates layers, changes their properties, and removes layers through prompts in the command bar.

LayersPanelClose

Closes the Layers panel.

Accessing the Command

command bar: *layerspanelclose*

: ***layerspanelclose***
The Layers panel is closed.

Command Options

No command options.

Related Commands

LayersPanelOpen - opens the Layers panel, which allows to edit the layers in the current drawing.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

LayersPanelOpen

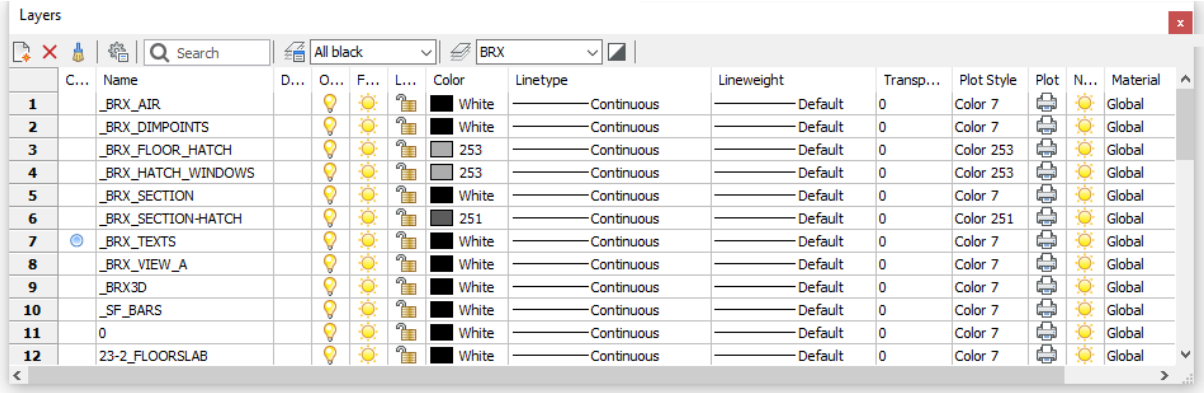
Opens the Layers panel, which allows to edit the layers in the current drawing, apply layer states and layer filters.

Accessing the Command

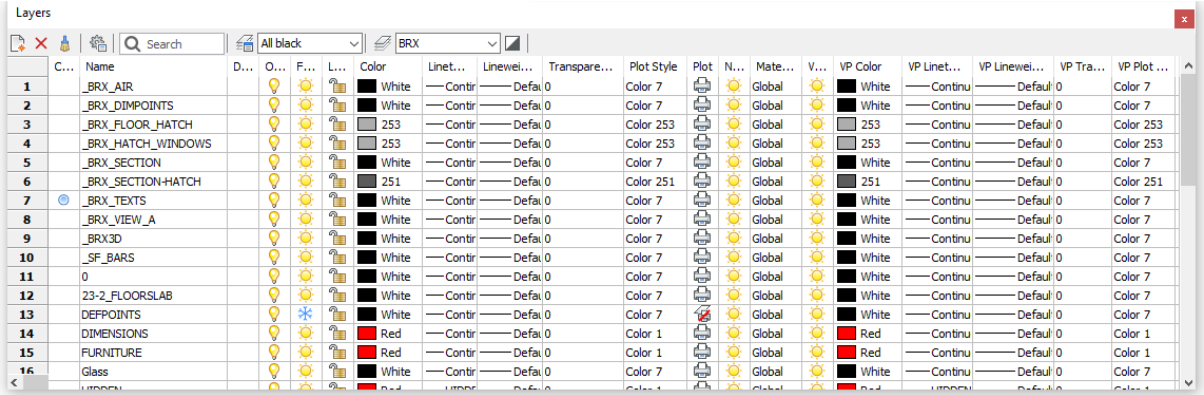
command bar: *layerspanelopen*

: ***layerspanelopen***

Displays a dialog box:



Layers dialog box in model space

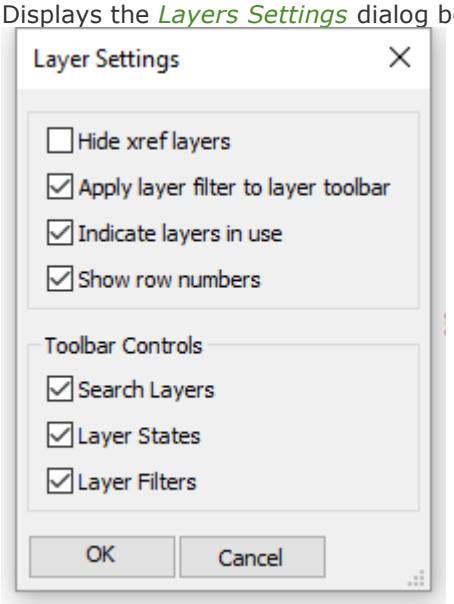


Layers dialog in paper space with the extra VP layer properties which apply to the current paper space layout or a viewport in the current layout.

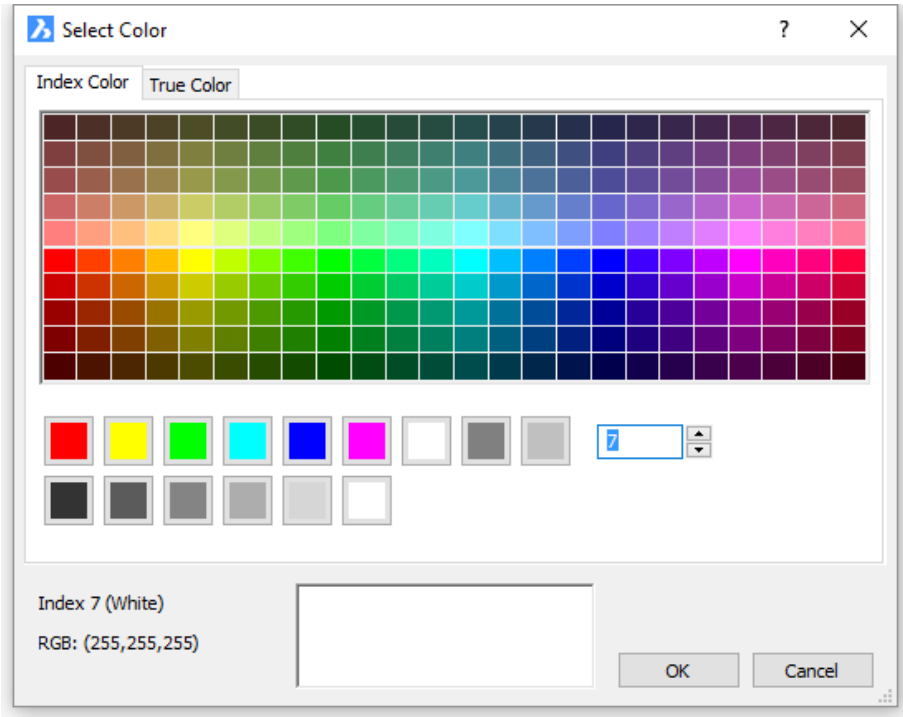
Command Options

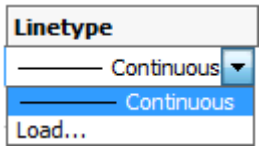
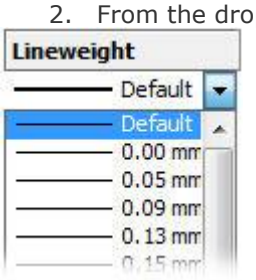
Option	Description
New	Creates new layers. A new layer is created as a copy of the currently selected layer. Keyboard shortcut: Ctrl+N
Delete	Deletes layers. The following layers cannot be deleted: • 0

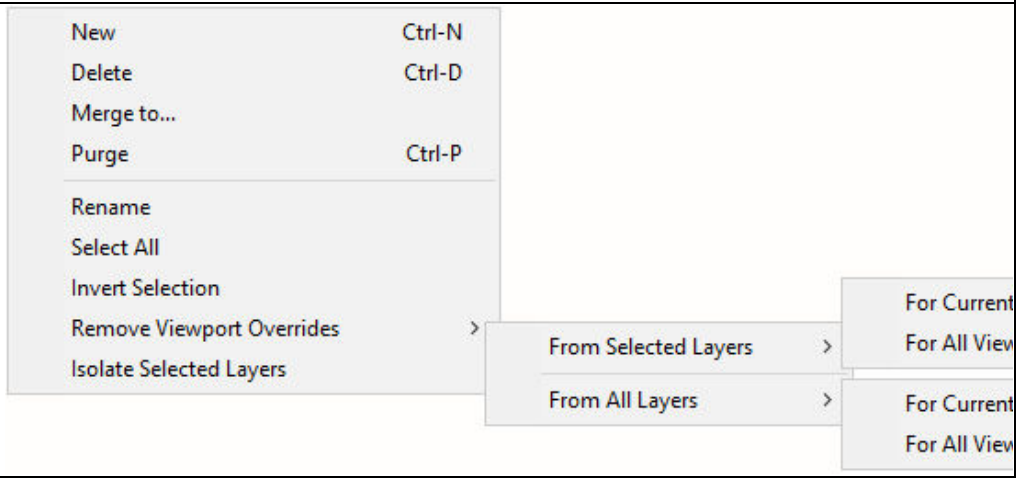
	• Any layer with entities. Keyboard shortcut: Ctrl+D

Purge	Purges layers from the current drawing. The following layers cannot be purged: • 0 • Current layer. • Any layer with entities. Keyboard shortcut: Ctrl+P
Settings	Displays the *Layers Settings* dialog box: 
Search	Allows to search for layers. Layer names containing the character sequence in the search field are filtered out. The layer list updates as you type. The *Search* field is available only if the *Search Layers* option on the Layer Settings dialog box is checked.
All black	Allows to select a Layer State. Click the field, then select a layer state in the list. The *Layer States* field is available only if the *Layer States* option on the Layer Settings dialog box is checked.
BRX	Allows to apply a layer filter. Click the field, then select a layer filter in the list. The *Layer filters* field is available only if the *Llayer Filters* option on the Layer Settings dialog box is checked.
	Allows to invert the current layer filter.
Current	Marks the current working layer. Only one layer can be current at a time. Click to *Current* field to make a layer current. Indicates the layer usage if the *Indicate layers in use* opyion is checked on the Layer Settings dialog box is checked. The following icons display in the *Layer Usage* column: • layer is current • layer is not empty

	• layer is empty When in a paper space layout or in a paper space viewport (model space with floating viewports) the following icons indicate VP overrides exist: • VP overrides exist and layer is current • VP overrides exist and layer is not empty • VP overrides exist and layer is empty
Layer Name	Specifies the name of the layer. • Click in the column, then type a character to jump to the first layer starting with that character. • To rename: 1. Click the name twice, slowly. 2. Enter the new layer name.
Description	(option) Describes the content of the layer. Click the field, then type.
On/Off	Toggles the selected layers between on and off: On - sets the normal state. Off - hides entities from view; entities are included in the calculation of extents. Click a lightbulb icon to change its status.
Freeze	Toggles the selected layers between frozen and thawed: Thaw - sets the normal state. Freeze - hides entities from view; entities are excluded in the calculation of extents, and are not exported with commands such as WBlock. Click an icon to toggle its status.
Locked	Toggles the selected layers between locked and unlocked: Unlocked - sets the normal state. Locked - prevents entities from being edited; entities are still visible. Click an icon to toggle its status.
Color	Specifies the color of the layer. Default color is white (black). To change the color: 1. Click a color square to see the Select Color dialog box.

	 2. Choose a color, and then click OK. See **Color** command.

Linetype	Specifies the linetype of selected layers. Default is "Continuous." To change the linetype: 1. Click a linetype sample. 2. Choose one from the droplist.  To load additional linetype definitions: 1. Click the Load option. 2. In the Load Linetypes dialog box, choose a linetype, and then click OK. See **Linetype** command.
Lineweight	Specifies the linetype of selected layers. Default is "Default." To change the lineweight: 1. Click the lineweight sample. 2. From the droplist, choose another linetweight.  See **LWeight** command.
Plot Style	Specifies the plot style of selected layers. Default is "Color_7." To change the plot style"

	1. Click a plot style name. 2. From the droplist, choose another style name. Plot styles are available only when they are enabled in the drawing; by default, they are not. See **PlotStyle** command.
Plot	Toggles the selected layers between plotable and not plotable: • Plot - plots layer. • No Plot - does not plot layer. Click an icon to toggle its status.
New Viewport	Specifies whether the layer is visible (thawed) or invisible (frozen) in a new layout viewport.
Material	Specifies the material of selected layers. Default is "Global." To change the material: • Click the Material field. • Click again to display the droplist, containing the available materials. • Click to select a material in the list.
Transparency	Sets the Transparency factor between 0 (fully opaque) and 90 (fully transparent). The TRANSPARENCYDISPLAY system variable controls the display of the transparency in the *2DWireframe* visual style. Rendered visual styles always display transparency
Context Menu	

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

-Layer - creates layers and changes their properties in the command bar.
Layer - creates layers and changes their properties in the Drawing Explorer.
LayersPanelClose - closes the Layers panel.
LayerState - saves and restores layer states.
LayMCur - changes the working layer to that of a selected entity
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.

LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.
Materials - Creates materials and edits their properties through the Drawing Explorer.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.
VpLayer - changes the properties of layers in the current paperspace viewport.

Explorer - Opens the Drawing Explorer dialog box.

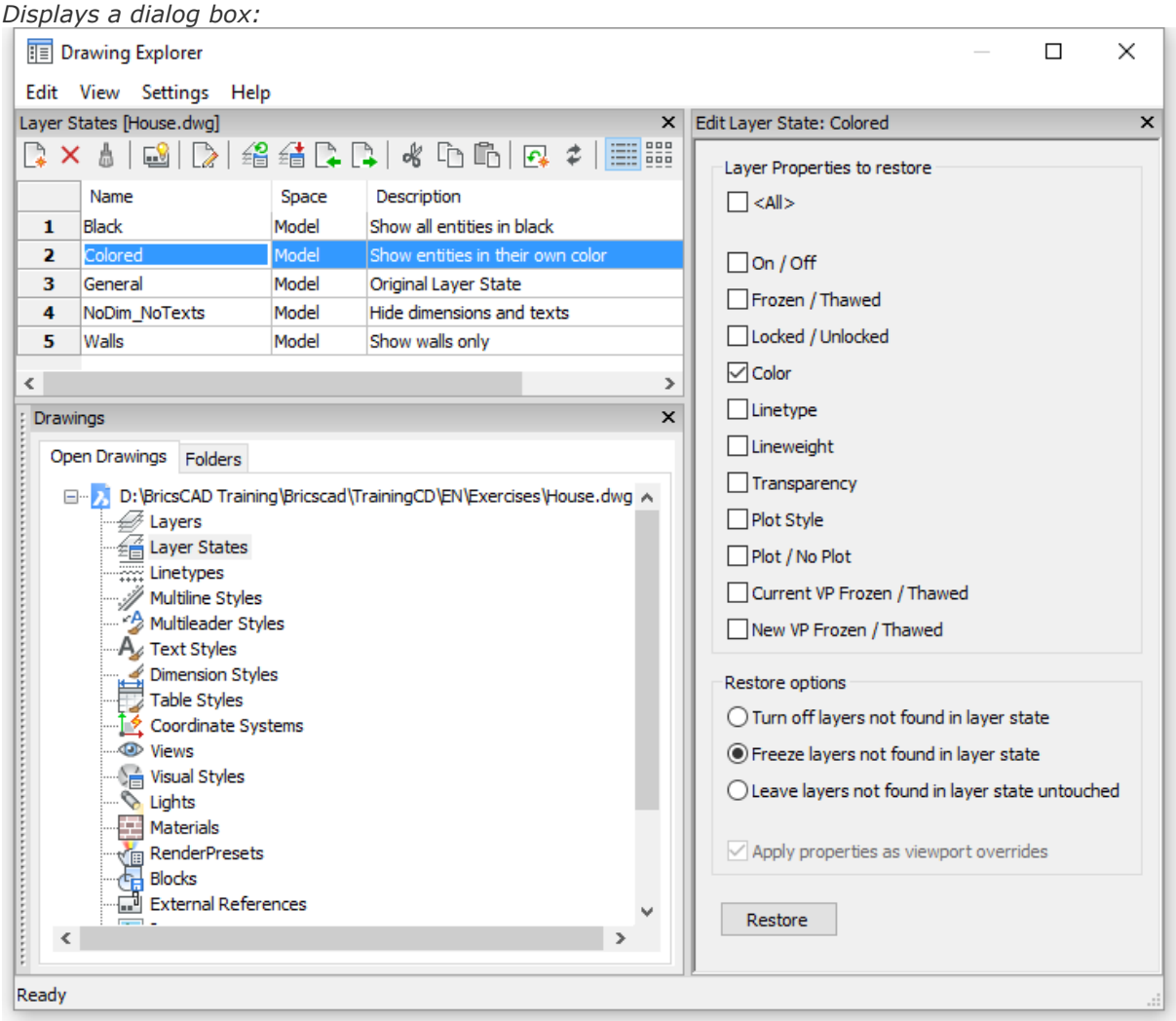
LayerState

Creates and controls layer states in the Drawing Explorer.

Accessing the Command

menu bar: *Settings | Layers | Layer States...* and *Tools | Drawing Explorer | Layer States...*
toolbar: *Layers |*
command bar: *layerstate*

: ***layerstate***
Displays a dialog box:



Create or choose a layer state, and then click X.

Press **F1** *to access help.*

Command Options

Option	Description
New	Creates a new layer state with the default name of NewLayerState*n*. Saves the current state of all layer properties. For example, the layer state records which layers are frozen, turned off, and locked.

Delete	Erases selected layer states.
Restore Restore	Restores the selected layer state. The layer properties change their state after closing the Drawing Explorer dialog box.
Overwrite	Overwrites (saves) the current state of layers.
On	Turns on all layer properties.
Off	Turns off all layer properties.
Import	Imports layer states from LAS files. Displays the Import Layer States dialog box. Choose an LAS file, and then click Open.
Export	Exports layer states to an LAS file. Displays the Export Layer States dialog box. Name the LAS file, and then click Save.
Name	Names the layer state. Click to rename the state.
Space	Reports which space the layer state was saved: Model - saved in model tab. Layout - saved in layout tab. You must start the *LayerState* command in either model or a layout tab.
Description	Describes the layer state.
Layer Properties to Restore	Specifies which layer properties are restored when the layer state is restored. Click the checkbox of a property to select or unselect it. Click the *<All>* checkbox to select or unselect all properties.
Restore Options	Allows to control what happens with layers not found in the layer state (= layers added after the layer state was created). The selected option applies to all layer states. The options are: • Turn off ( ). • Freeze ( ). • Do nothing.
Apply properties as viewport overrides	Applies the layer state as viewport overrides (VP layer properties) to the current layout viewport. This option is available only on a layout within a layout viewport.

Procedures

To save the current layer state
To edit a layer state
To restore a layer state
To copy layer states to another drawing

To export a layer state
To import a layer state

Related Commands

Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

Explorer - Opens the Drawing Explorer dialog box.

LayMCur

Changes the working layer to that of a selected entity (short for "layer make current").

Accessing the Command

command bar: *laymcur*
alias: *setlayer*
menu bar: *Settings | Layers | Set Layer by Entity*
toolbar: *Entity Properties |*

: ***laymcur***
Prompts you in the command bar:
Select entity to set the current layer: *(Choose an entity.)*

Command Options

No Options.

Related Commands

Layer - creates layers and changes their properties through a dialog box.
-Layer - creates layers and changes their properties through the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.
LayOff - turns off layers associated with selected entities.
LayOn - turns on all layers that have been turned off.

LayOff

Turns off layers associated with entities selected in the drawing; off layers cannot be seen (short for "layer off").

Accessing the Command

command bar: *layoff*
menu bar: *Settings | Layers | Layer Off*
toolbar: *Layers |*

: ***layoff***
Prompts you in the command bar:

Current setting: Viewports=Off, Block nesting level=Block

Select an entity on the layer to be turned off or [Settings]: *(Choose an entity, or type S.)*
This layer has been turned off: <LayerName>.
Select an entity on the layer to be turned off or [Settings]: *(Choose another entity, or press Enter to exit the command.)*

Command Options

Option	Description
Select an entity on the layer to be turned off	Selects an entity. Choose an entity whose layer should be turned off. This prompt repeats itself until you press Enter; this allows you to turn off more than one layer.
Settings	Specifies how viewports, blocks, and xrefs should be turned off; prompts you: **Enter setting type for [Viewports/Block selection]** - type V or B.
Viewports	Specifies how layers and viewports should be turned off; prompts you: **In paper space viewports use [Vpfreeze/Off]** - type V or O. • **Vpfreeze** - freezes the layers in the current viewport. • **Off** - turns off layers (made invisible).
Block selection	Specifies how blocks and xrefs should be frozen; prompts you: **Enter Block Selection nesting level [Block/Entity/None]** - type B, E, or N. • **Block** - freezes layers associated with the block insertion or xref attachment (default). • **Entity** - freezes layers associated with entities selected in the block or xref. (Entities that make up blocks can sometimes be on different layers.) • **None** - does not freeze the block or xref.

Related Commands

LayOn - turns on all layers that have been turned off.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.

LayOn

Turns on all off layers (short for "layer off").

Accessing the Command

command bar: *layon*
menu bar: *Settings | Layers | Layer On*
toolbar: *Layers |*

: ***layon***
BricsCAD responds in the command bar:
All layers have been turned on.

This command turns ON all layers, whether turned off by the LayOff or by other commands. Frozen layers are not affected.

Command Options

No options.

Related Commands

LayOff - turns off layers associated with selected entities.
Layer - creates layers and changes their properties in the Drawing Explorer.
-Layer - creates layers and changes their properties in the command bar.
LayerState - saves and restores layer states.
LayFrz - freezes layers associated with selected entities.
LayThw - thaws all frozen layers.
LayIso - isolates layers associated with selected entities.
LayLck - locks layers associated with selected entities.
LayUlk - unlocks layers associated with selected entities.

Layout

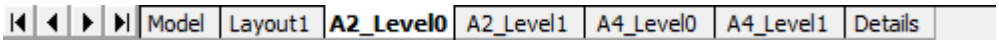
Creates, copies, renames, and deletes layouts.

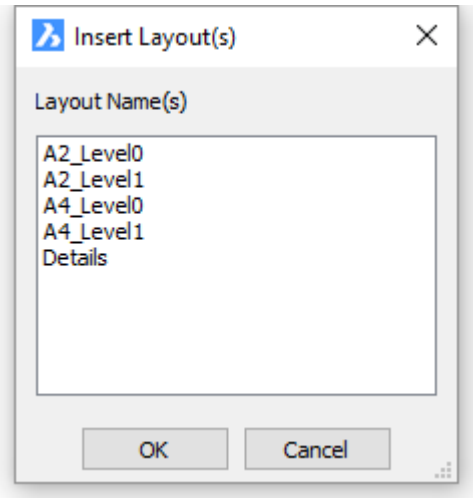
Accessing the Command

command bar: *layout*
menu bar: *Insert | Layout*
toolbar: *Insert* | and *Layout* |

: ***layout***
Prompts you in the command bar:
Enter layout option [? to list/Set/New/Copy/Delete/Rename/SAve/Template/neXt/Previous] <Set>:
(Enter an option.)

Command Options

Option	Description
? to list	Lists the names of layouts already defined in this drawing.
Set	Switches to the specified layout: Enter a layout name - enter a name of a layout. BricsCAD switches to that layout.
New	Creates new layouts; prompts you: Enter new layout name - specify the name of the new layout. The new layout is added as a tab.
Copy	Makes a copy of an existing layout; prompts you: Enter the name of the layout to copy: <layout1> - enter a name of an existing layout. Enter new layout name - specify the name of the new layout. The new layout is added as a tab. 
Delete	Erases layouts from the drawing; prompts you: Enter a layout name - specify the name of the layout to be removed.
Rename	Renames layouts; prompts you: Enter a layout name - enter a name of an existing layout. Enter new layout name - specify the name of the new layout.
SAve	Saves layouts as DWG files: Enter a layout name - enter a name of a layout. Displays the Create Drawing File dialog box. Enter a name for the drawing, and then click Save. The DWG file contains the saved layout, as well as the default ones: Model, Layout 1, and Layout2.
Template	Imports layouts from DWG drawing files. Displays the Select Template From File dialog box. Choose a DWG file, and then click Open. Displays the Insert Layout(s) dialog box:

	 Choose one or more layout names,and then click OK. If the layout name already exists, BricsCAD appends its name. For example, if the imported layout is named "Layout 1,"then BricsCAD names the new layout "Layout3-Layout1."
Next	Displays the next layout.
Previous	Displays the previous layout.

Status Bar Menu

Right-click TILE on the status bar for the shortcut menu:

Shortcut Menu	Description
Layout1	Switches the drawing editor to Layout1.
Model	Switches the drawing editor to Layout1.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Procedures

Creating a new layout
Copying layouts
Importing layouts
Renaming layouts
Arranging the layout tabs
Deleting a layout

Related Commands

LayoutManager - manages layouts through a dialog box.
MSpace - switches to model space.
PsetupIn - imports page setup definitions from another drawing through a dialog box.
-PsetupIn - imports page setup definitions from another drawing through the command bar.
PSpace - switches to paper space.
MView - creates multiple viewports in paper space.
VPorts - creates multiple viewports in model space.

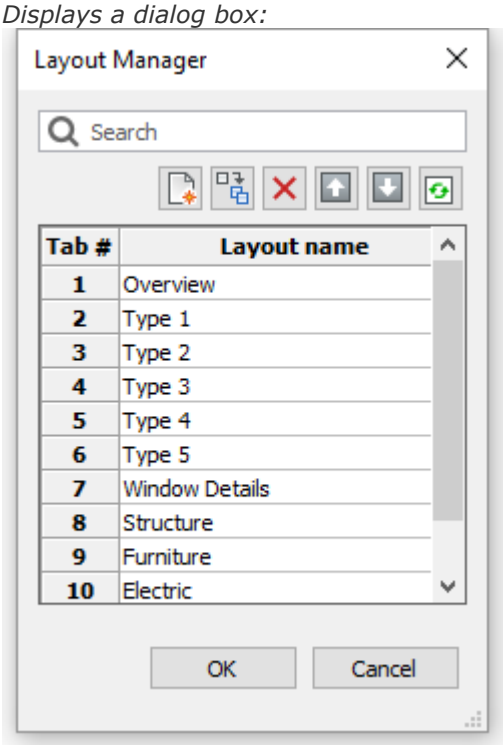
LayoutManager

Manages layouts through a dialog box.

Accessing the Command

command bar: *layoutmanager*
graphic screen:

: ***layoutmanager***
Displays a dialog box:



Command Options

Option	Description
	Creates a new layout. Type a name in the *Layout name* field to replace the default Layoutxx name.
	Copies the selected layou(s)t.
	Deletes the currently selected layout(s).
	Moves the currently selected layout(s) up in the list (= moves the layout tabs to the left).
	Moves the currently selected layout(s) down in the list (= moves the layout tabs to the right).
	Clears the current selection.
Search	Allow to filter the layout list. Type a search string in the *Search* field. Only layouts which contain the search string are listed.

Related Commands

Layout - creates, copies, renames, and deletes layouts.

Leader

Draws leaders.

Accessing the Command

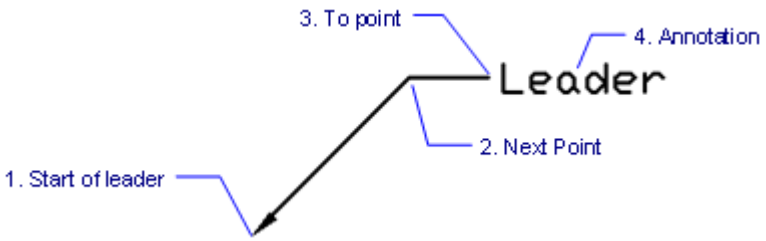
command bar: *leader*
aliases: *le, lead*

: ***leader***
Prompts you in the command bar:

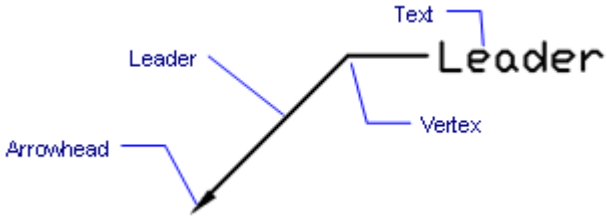
Start of leader: **(Pick a point.)**

Next point: **(Pick a point.)**

To point [Format/Undo/Annotation] <Annotation>: **(Pick a point, or enter an option.)**



Leaders consist of the following parts:



Command Options

Option	Description
To point	Specifies the next vertex. Pick a point or press Enter to stop adding vertices and start the Annotation option.
Format	Specifies the leader style; prompts you: [Arrow/None/SPline/STraight/Exit] <Exit> - enter an option. • Arrow - Draws an arrowhead at the leader's start point (default). • None - Draws no arrowhead. • Spline - Draws the leader as a spline. • Straight - Draws the leader as straight line segments (default). • Exit - Returns to the previous prompt.
Undo	Undoes the last leader line segment.
Annotation	Starts adding text to the end of the leader line. The annotation is created as an Mtext entity; prompts you:

	First line of annotation text <options> - Press Enter to choose Options or type the first line of text, then press Enter. Next line of annotation text - Press Enter to stop adding text or type the next line, then press Enter. Caution: the annotation is independent of the leader line; when moving a leader, be sure to include the annotation in the selection set.
Options	Specifies the type of annotation; prompts you: Dimension text options [Block/Copy/None/Tolerance/Mtext] <Mtext> - enter an option. • Block - Uses a block as the annotation; displays prompts identical to that of the -Insert command: • Copy - Uses any other text entity in the drawing as the annotation; prompts you: **Select an entity** - select a Text or Mtext entity. Text entities are converted to Mtext. • None - Creates a leader without annotation. • Tolerance - Uses tolerance notation for the annotation; displays the Geometric Tolerance dialog box. • Mtext - Displays a Text Formatting toolbar and the Mtext entry field.

Related Commands

DimLeader - places leaders with text, blocks, entities, or tolerances.
QLeader - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.
MLeader - creates multileader entities using the current multileader style.

DimLinear - places linear dimensions in drawings: horizontal, vertical, and rotated.
DimAligned - places aligned dimensions.
DimBaseline - places additional dimensions from a common baseline.
DimContinue - continues dimensions.
DimDiameter - places diameter dimensions.
DimRadius - places radius dimensions.
DimCenter - places center marks.
DimAngular - places angular dimensions.
DimOrdinate - places x and y ordinate dimensions.
Tolerance - places tolerance tags in drawings.
Dim - places and edits dimensions at the 'Dimensioning command:' prompt.
Dim1 - places and edits a single dimension at the 'Dimensioning command:' prompt.

DimEdit - changes the wording and angle of dimension text; changes the angle of extension lines.
DimTEdit - changes the position of dimension text.
DdEdit - edits dimension text.
Properties - edits the properties of dimensions through a palette.

DimStyle - creates and edits dimension styles through the Drawing Explorer.
-DimStyle - saves and applies dimensions styles at the command line.
Style - sets the text style used by dimensions.
DimOverride - overrides the values of the current dimension style.

Lengthen

Changes the length of open objects, such as lines and arcs.

Accessing the Command

command bar: *lengthen*
aliases: *len, editlen*
menu bar: *Modify | Lengthen*
toolbar: *Modify |*

: ***lengthen***
Prompts you in the command bar:
Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>: *(Enter an option, or choose an entity.)*

Command Options

Option	Description
Select entity to list length	Reports the current length of an open entity. Reports: Current length - reports the length in current units.
DYnamic	Changes the length interactively; prompts you: Mode/<Select entity to change> - enter an option.
Select entity to change	Changes the length of the selected entity; prompts you: New end point for line - drag the line longer or shorter.
Mode	Switches lengthening mode; returns to original prompt: **Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length> - enter an option.**
Increment	Changes the angle and length by a specified amount; prompts you: Angle/<Enter length increment (0.0000)> - enter an option.
Enter length increment	Specifies the length by which to lengthen or shorten the entity: • Lengthen - enter a positive number. • Shorten - enter a negative number.
Angle	Specifies the angle by which to lengthen the entity; prompts you: Enter angle increment - enter an angle. This option applies only to arcs.
Percent	Changes the length of entities by a percentage; prompts you: **Enter percent length <0.0000> - enter a number.** This option lengthens or shortens to the percentage, not by the percentage. For example, enter 25 (for 25%), and a 1"-long line is shorted to 0.25". Enter 150%, and the 1" line becomes 1.5".
Total	Changes the length to the amount specified; prompts you: **Angle/<Enter total length (0.0000)> - enter an option.**

Enter total length	Specifies the length to which to lengthen or shorten the entity. For example, enter 2 and a 1"-long line is lengthened to 2".

Angle	Specifies the angle to which to size the entity; prompts you: Enter angle increment - enter an angle. This option applies only to arcs. For example, enter 45 and a 180-degree arc is shortened to 45 degrees.

Procedures

Change the length of an entity dynamically
Modify the included angle of an arc

Related Commands

Extend - lengthens open entities.
Stretch - stretches entities longer and shorter.
Change - changes the length of lines.
Trim - shortens entities.

LicenseManager

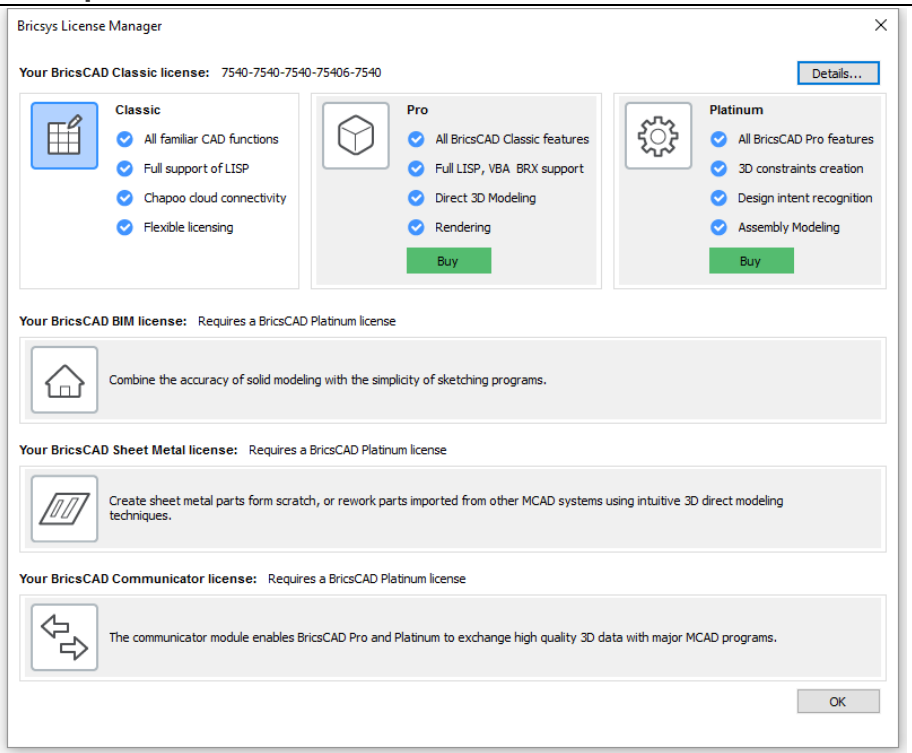
Displays the License Manager dialog, summarizing your available licenses (BricsCAD / BIM / Sheet Metal / Communicator).

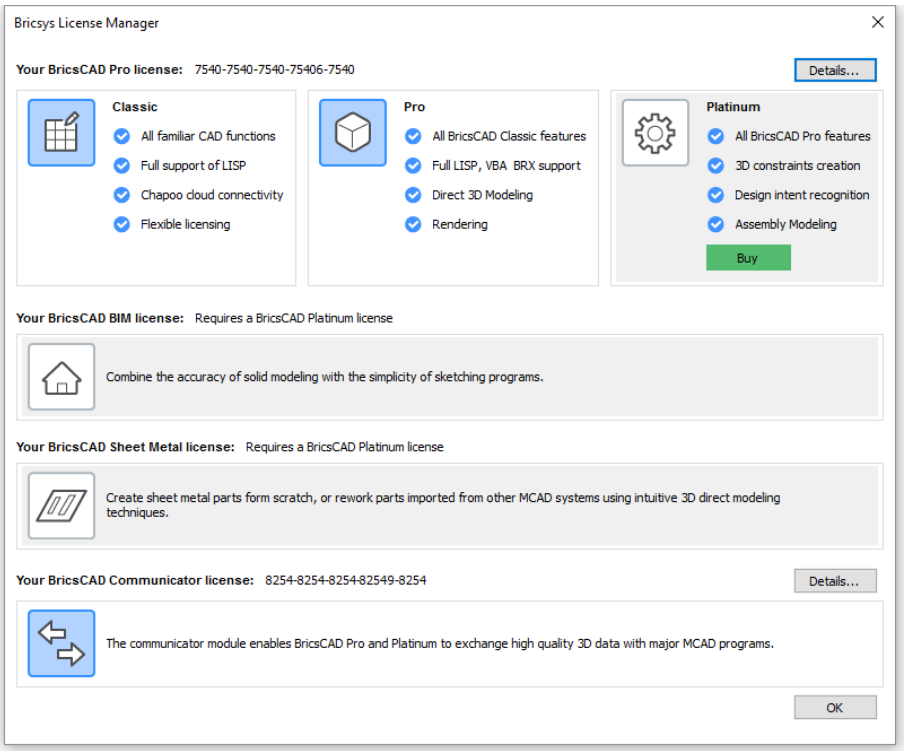
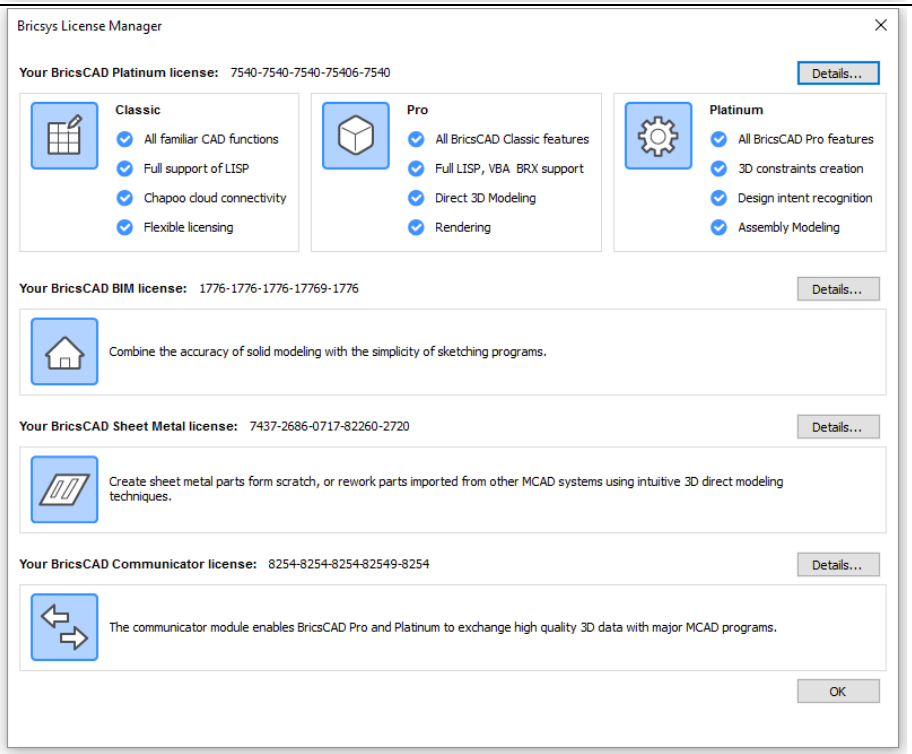
Accessing the Command

command bar: *licensemanager*
menu bar: *Help | License Manager*

: ***licensemanager***
Displays the Bricsys License Manager dialog box.

Command Options

Option	Description
Classic License	

Pro License	
Platinum License	

Related Commands

LicEnterKey - enters the license key number.
LicProperties - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key.
LicPropertiesBim - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key.
LicPropertiesCommunicator - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.
LicPropertiesSheetMetal - reports the BricsCAD Sheet Metal license information in a dialog box. Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.
RunAsLevel - allows to run BricsCAD in a lower license level.

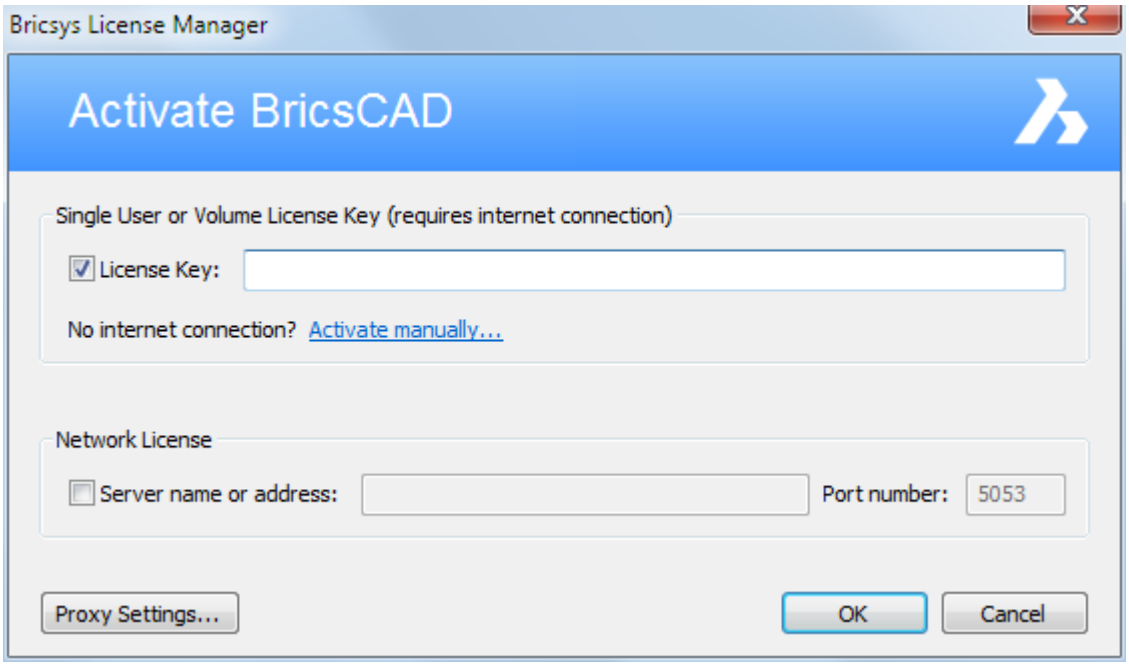
LicEnterKey

Enters the license key number (short for "licence enter key").

Accessing the Command

command bar: *licenterkey*
transparent: *'licenterkey*

: ***licenterkey***
Displays a dialog box:



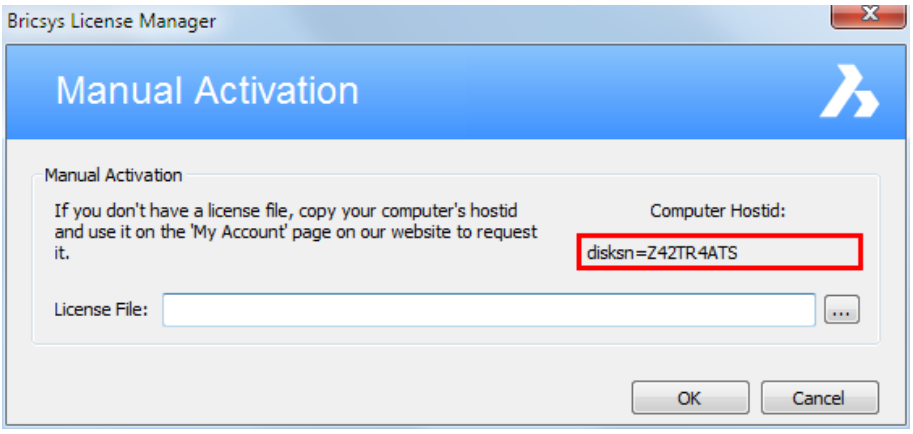
Enter your license information, and then click OK.

Use this command only to change the license number. To view the license number, use the **LicProperties** command.

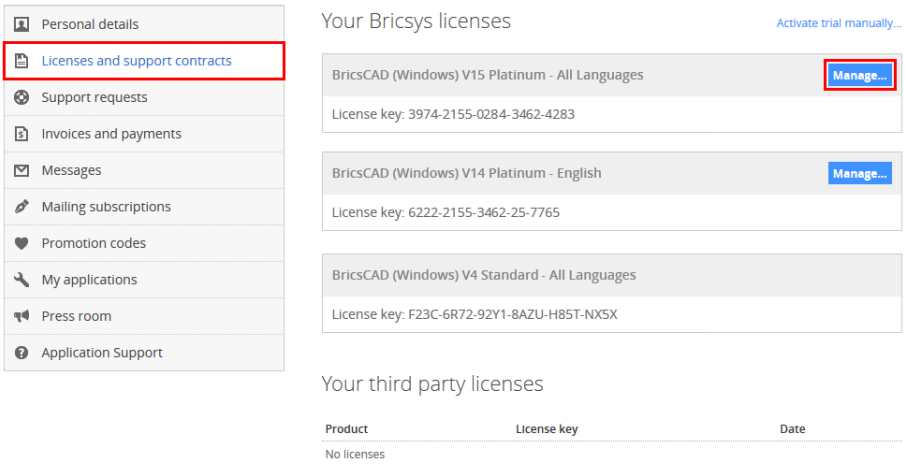
A separate license is required for the Classic, Pro and Platinum versions of BricsCAD.

Command Options

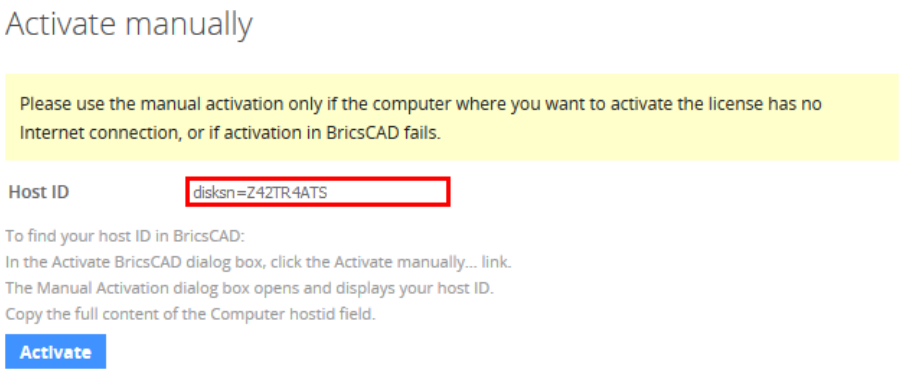
Option	Description
License Key	Allows to activate a single user or volume license key. 1. Enter a license key in the *License Key* field. 2. Press the *OK* Button. The license is activated over the internet. A license file is created.
Activate manually...	Allows to activate a single user or volume license key on a computer that is not connected to the internet. Another computer, with internet connection, is needed to retrieve a valid license file from the BricsCAD license server.



1. Logon to your account on the Bricsys website and select *Licenses and Support Contracts*.

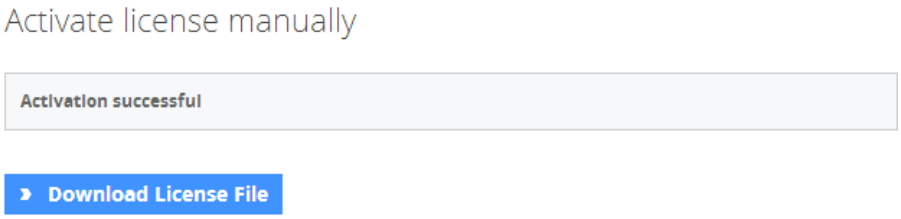


2. Click the *Manage...* button of the license you want to activate.
3. Copy/paste the computer host ID of the computer you want to install BricsCAD on in the *Host ID* field.

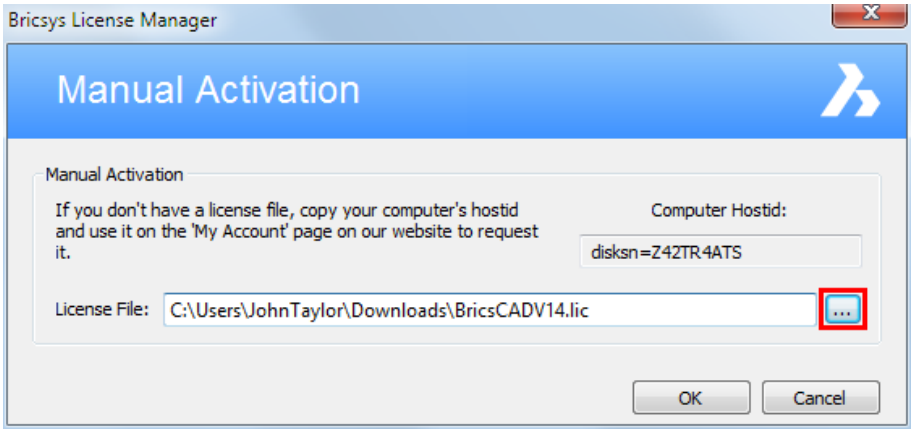


4. Click the *Activate* button.

5. A license file is generated and made available for download.

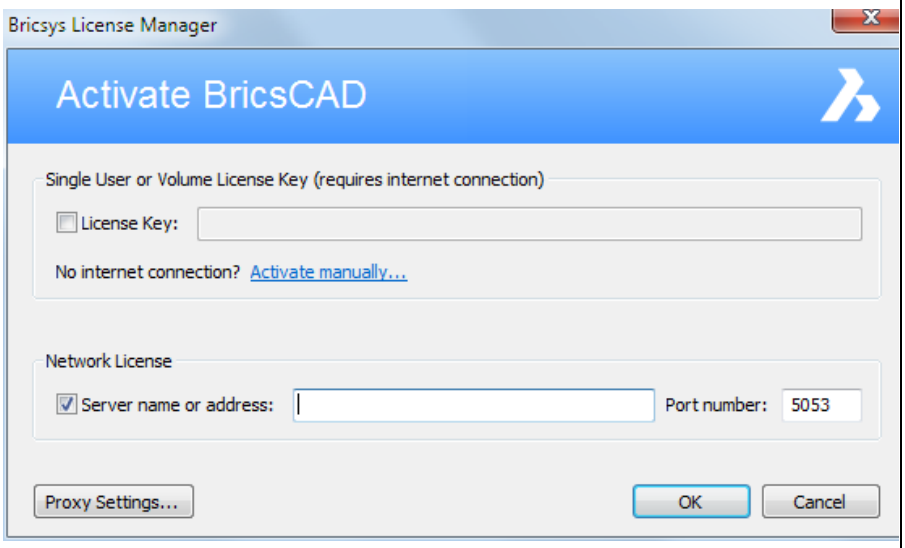


6. Click the *Download License File* button to download the *BricsCADVxx.lic* file.
7. Copy the license file to the computer you want to install BricsCAD on.
8. Click the *Browse* button (...) on the *Bricsys License Manager - Manual Activation* dialog box and select the license file.

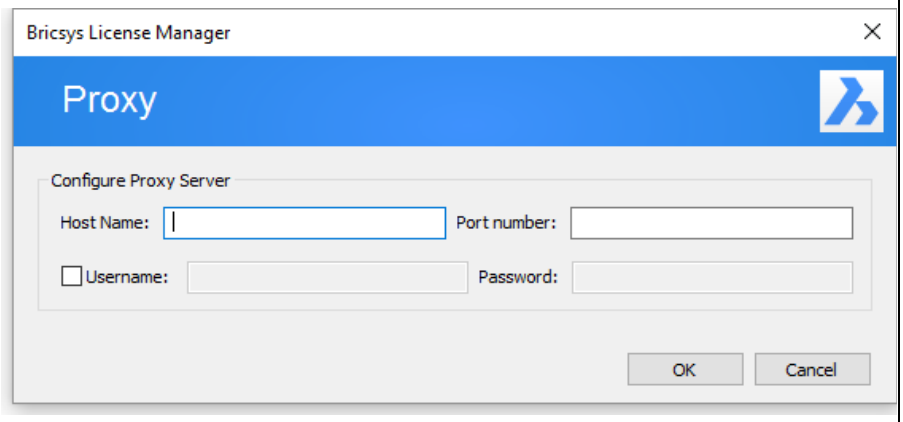


9. Click the *OK* button.

Network License	Allows to retrieve a network license from a license server.



1. Type the host name or IP address of the license server in the *Server name or address* field.
2. Type the port number in the *Port Number* field.
3. Press enter or click the *OK* button.

Proxy Settings...	Allows to specify proxy settings if your computer connects to the internet or a local network through a proxy server. 

Related Commands

Status - reports on the state of the drawing.
LicProperties - reports the license number and owner.
LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal)

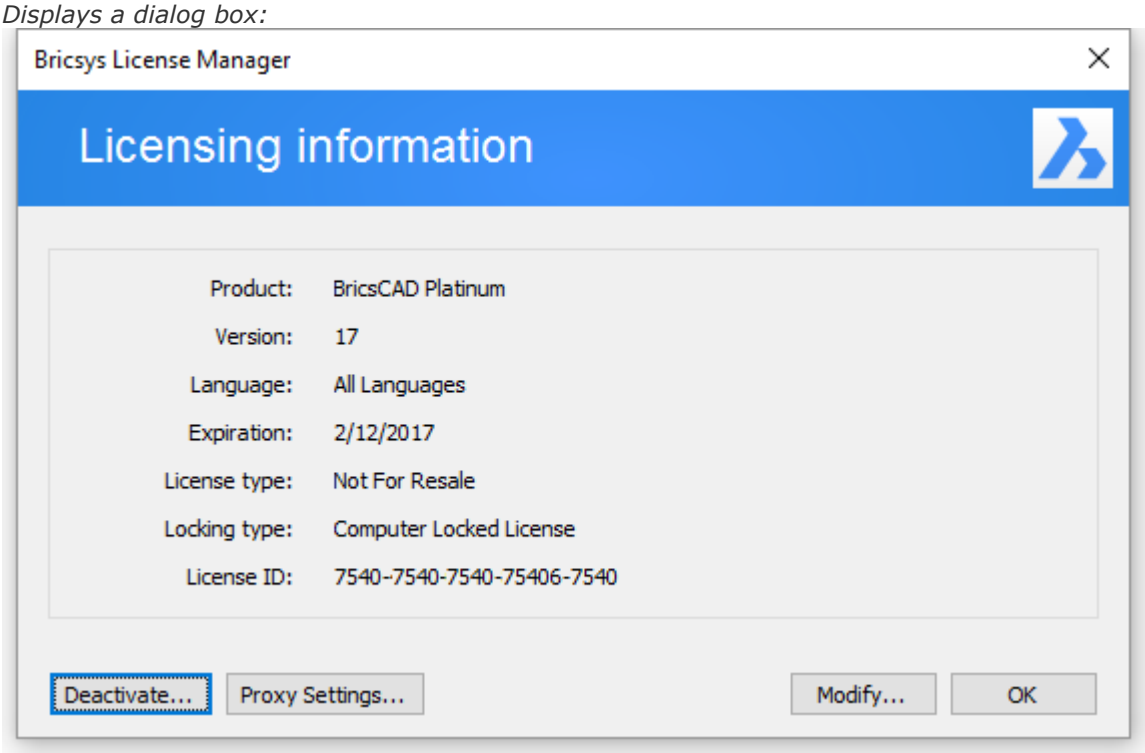
LicProperties

Reports the BricsCAD license information in a dialog box.
Allows to modify or deactivate a BricsCAD single user or volume license key.

Accessing the Command

command bar: *licproperties*

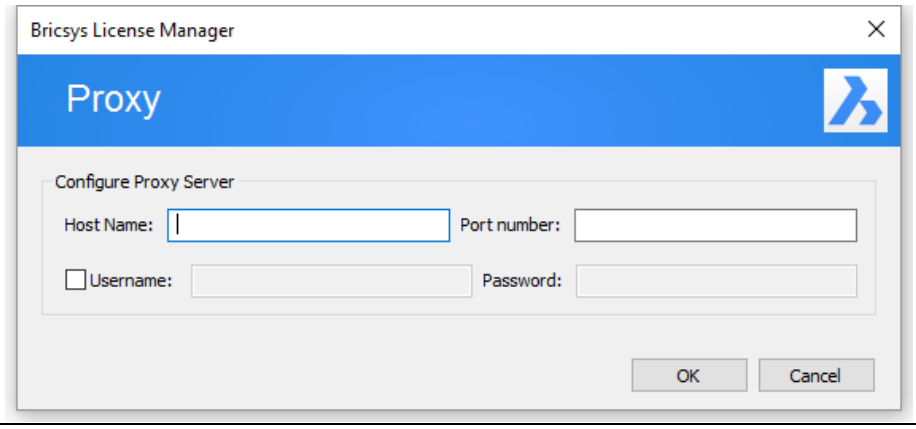
: ***licproperties***
Displays a dialog box:



*Review license information, and then click **OK**.*

Command Options

Option	Description
Deactivate...	Deactivates the license key. BricsCAD can no longer be used on this computer. The license key is available for activation on another computer. An internet connection is needed to deactivate a license key. You must deactivate a license key: • To migrate a license key to a different computer. • When removing BricsCAD from a computer permanently. • Before installing a new hard disk, changing partitions or upgrading the operating system, e.g. from Windows 7 to 8.
Proxy Settings	Allows to specify proxy settings if your computer connects to the internet through a proxy server.

	
Modify	Displays the *Activate BricsCAD* dialog box. See LicEnterKey command.

Related Commands

Status - reports on the state of the drawing.
LicEnterKey - changes the license number and owner.
LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal)
LicPropertiesBim - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key.
LicPropertiesCommunicator - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.
LicPropertiesSheetMetal - reports the BricsCAD Sheet Metal license information in a dialog box. Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.

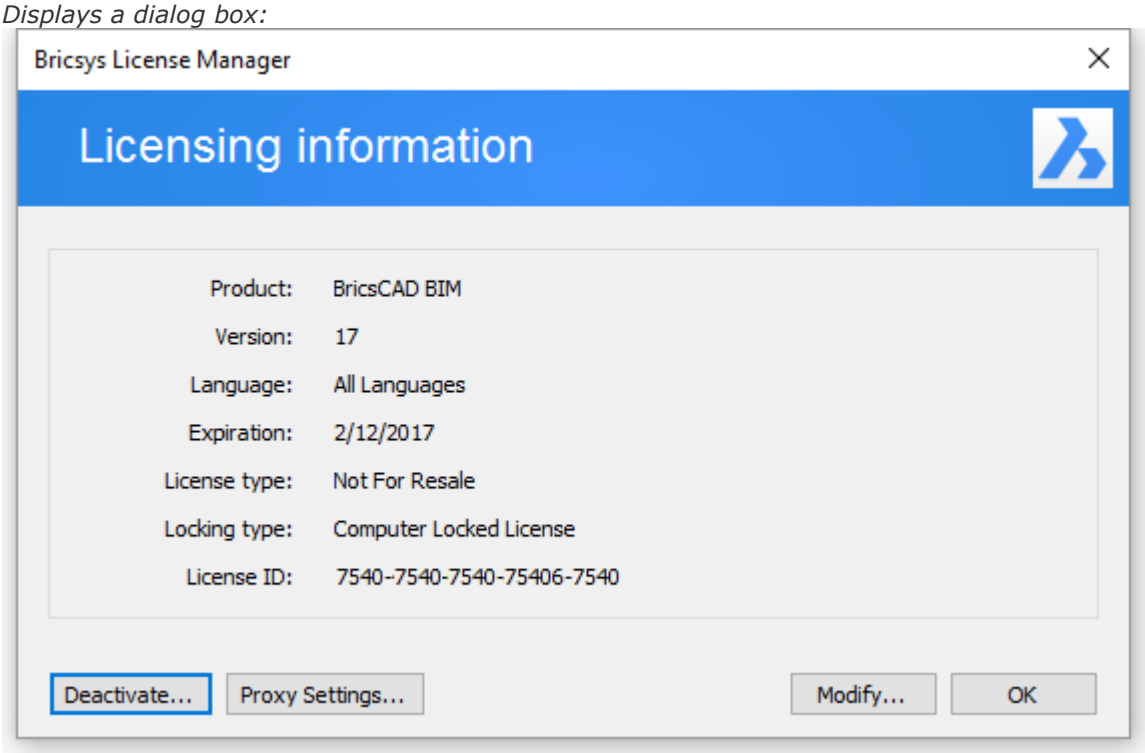
LicPropertiesBim

Reports the BricsCAD BIM license information in a dialog box.
Allows to modify or deactivate a BricsCAD BIM single user or volume license key.

Accessing the Command

command bar: *licpropertiesbim*

: ***licpropertiesbim***
Displays a dialog box:



*Review license information, and then click **OK**.*

Command Options

See LicProperties command.

Related Commands

Status - reports on the state of the drawing.
LicEnterKey - changes the license number and owner.
LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal)
LicProperties - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key.
LicPropertiesCommunicator - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.
LicPropertiesSheetMetal - reports the BricsCAD Sheet Metal license information in a dialog box. Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.

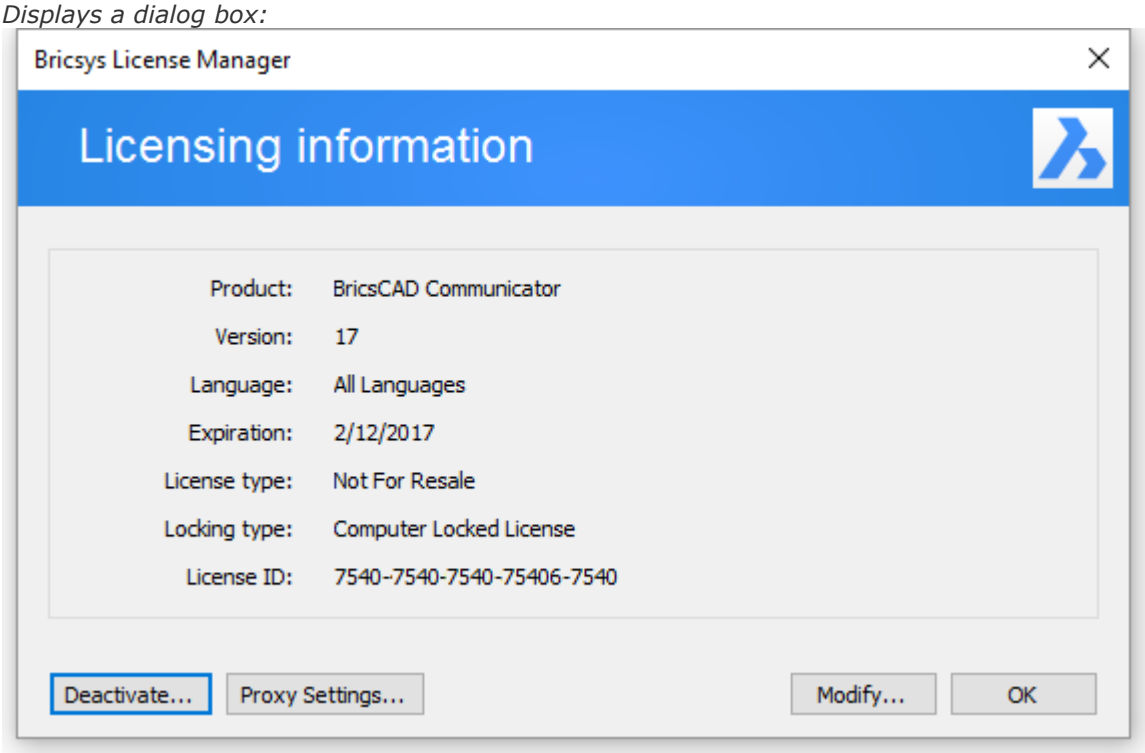
LicPropertiesCommunicator

Reports the BricsCAD Communicator license information in a dialog box.
Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.

Accessing the Command

command bar: *licpropertiescommunicator*

: ***licpropertiescommunicator***
Displays a dialog box:



*Review license information, and then click **OK**.*

Command Options

See LicProperties command.

Related Commands

Status - reports on the state of the drawing.
LicEnterKey - changes the license number and owner.
LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal)
LicProperties - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key.
LicPropertiesBim - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key.
LicPropertiesSheetMetal - reports the BricsCAD Sheet Metal license information in a dialog box. Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.

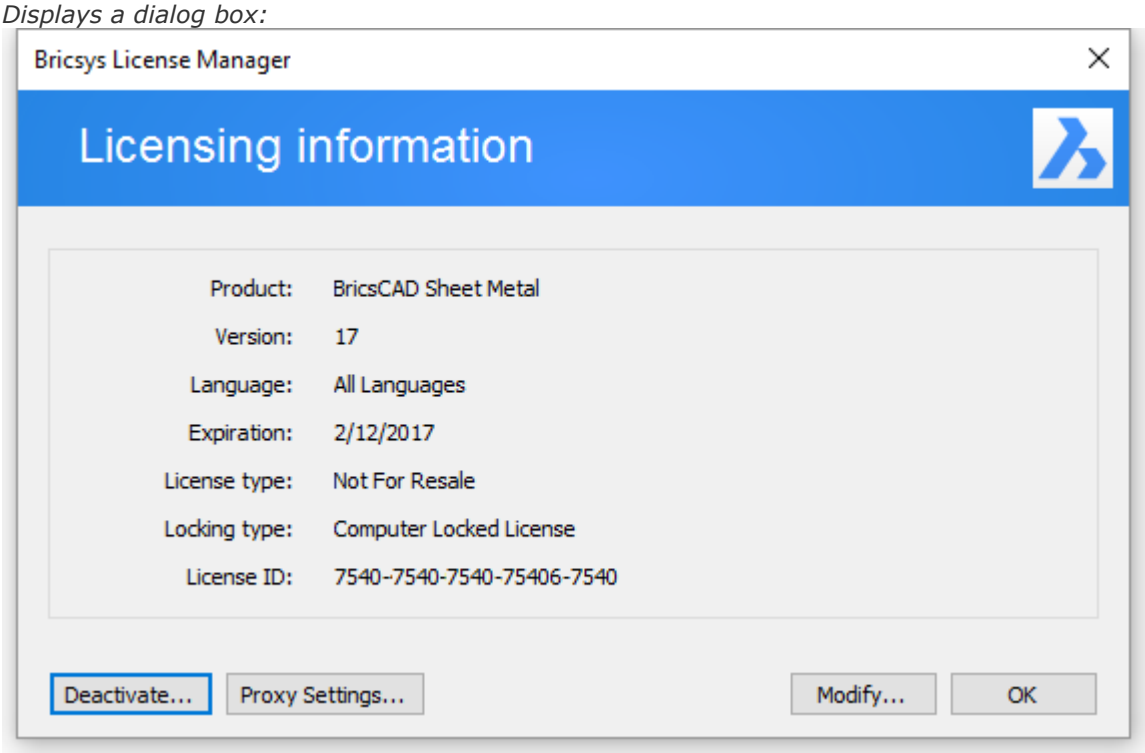
LicPropertiesSheetMetal

Reports the BricsCAD Sheet Metal license information in a dialog box.
Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.

Accessing the Command

command bar: *licpropertiessheetmetal*

: ***licpropertiessheetmetal***
Displays a dialog box:



*Review license information, and then click **OK**.*

Command Options

See LicProperties command.

Related Commands

Status - reports on the state of the drawing.
LicEnterKey - changes the license number and owner.
LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal)
LicProperties - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key.
LicPropertiesBim - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key.
LicPropertiesCommunicator - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.

Light

Creates a new light in the drawing.

Accessing the Command

command bar: *light*
toolbar: *Drawing Explorer - Lights |*

: ***light***
Prompts you in the command bar:
Enter light type [Point/Spot/Web/Distant] <Spot>: *(Enter an option or press Enter to create a spot light).*

Command Options

Option	Description
Point	Creates a point light. See POINTLIGHT.
Spot	Creates a spot light. See SPOTLIGHT.
Web	Creates a web light. See WEBLIGHT.
Distant	Creates a distant light. See DISTANTLIGHT.

Related Commands

Distantlight - creates a distant light.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Materials - creates materials and edits their properties through the Drawing Explorer.
Pointlight - creates a point light.
Render - calculates a photorealistic image of the 3D model.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
Weblight - creates a web light.

Lightlist

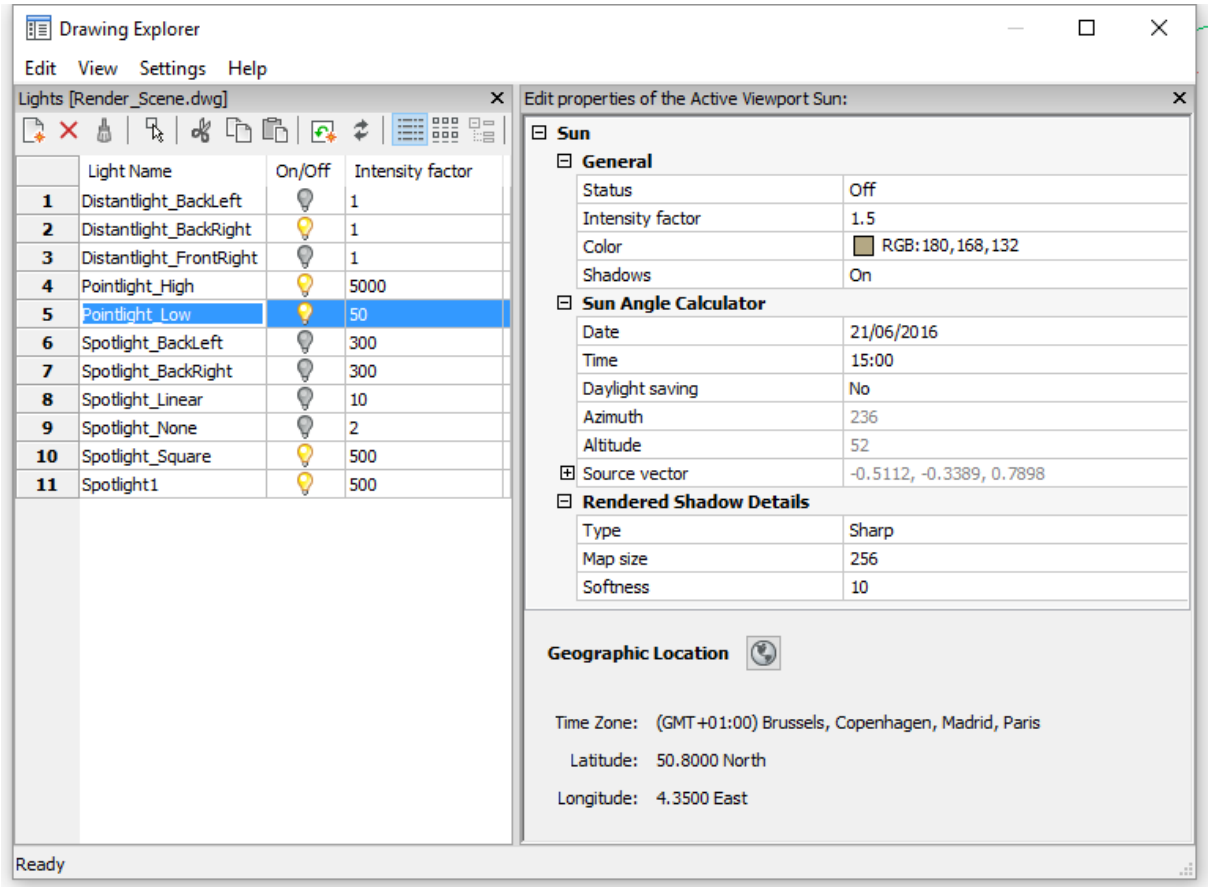
Displays the Drawing Explorer - Lights dialog window.

Accessing the Command

command bar: *lightlist*
alias: *LL*
menu bar: *Tools | Drawing Explorer | Lights...*
toolbar: *Lights |*

: ***lightlist***

Displays a dialog box:



Command Options

Option	Description
New	Starts the LIGHT command.
Delete	Deletes the selected lights.
Select in drawing	Closes the *Drawing Explorer - Lights* dialog box. Selects the light's glyph in the drawing; the light's properties display in the *Properties Bar*.
Rename	Edit the name of the light.

Select all	Selects all lights in the light list. Choose Select in drawing to select the glyphs of all lights in the drawing.
ON	Indicates the light is currently switched on. Click to turn the light off.
OFF	Indicates the light is currently switched off. Click to turn the light on.
Sun	Sets the properties of the sunlight. See the SunProperties command.
Geographic Location	Launches the GeographicLocation command.

Related Commands

Light: Creates a new light.
Render: Calculates a photorealistic image of the 3D model.
Sunproperties: Displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
GeographicLocation: Sets the geographic location of the drawing.
Explorer - Opens the Drawing Explorer dialog box.

Limits

Limits the extent of the drawing and the grid.

Accessing the Command

command bar: *limits*
transparent: *'limits*
menu bar: *Settings | Drawing Limits*

: ***limits***
Prompts you in the command bar:

Limits are off: Set lower left corner or [turn limits ON] <0,0>:

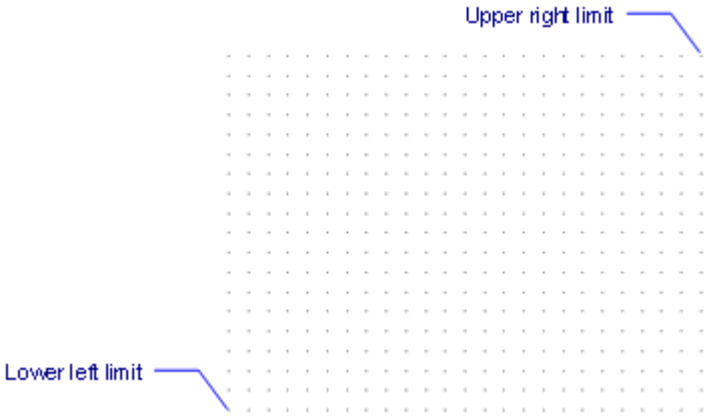
or

Limits are on: Set lower left corner or [turn limits OFf] <0,0>: (Enter x,y coordinates, pick a point or choose an option.)

Upper right corner <420,297>: (Enter x,y coordinates or pick a point.)

When limits are on, they prohibit drawing outside of the area defined by this command.

BricsCAD reports, "Selected point is outside limits. Please select another point."



The area also defines the extent of the grid display, whether limits are turned on or off, if the *Display beyond limits area* flag of the GRIDDISPLAY system variable is not set.

Command Options

Option	Description
ON	Turns on limits.
OFF	Turns off limits.
Lower left corner	Specifies the lower left corner of the limits.
Upper right corner	Specifies the upper right corner of the limits.

Related Commands

Grid - uses the limits to specify the extent of the grid display.
Plot and Print - plot to the limits of the drawing.
Zoom - zooms to the limits (or extents) with the All option.

Line

Draws line segments.

Accessing the Command

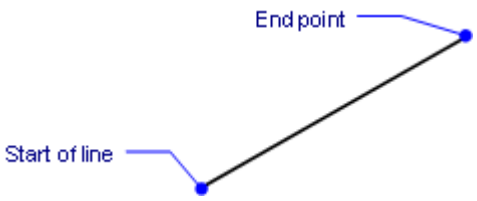
command bar: *line*
aliases: *l, 3dline*
menu bar: *Draw | Line*
toolbar: *Draw |*

: ***line***
Prompts you in the command bar:
Start of line or [Follow] <Last point>: (Pick a point, enter x,y,z coordinates, or choose an option.)

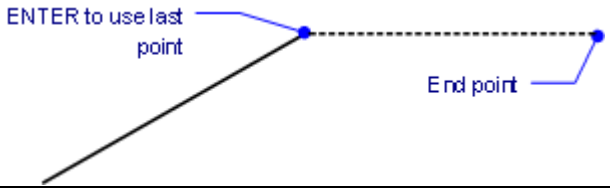
Set end point or [Angle/Length/Undo]: (Pick a point, enter x,y,z coordinates, or choose an option.)

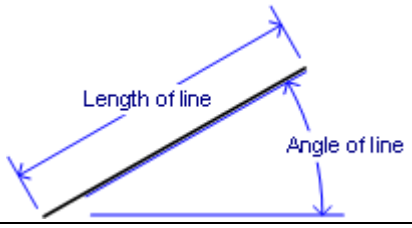
Set end point or [Angle/Length/Follow/Undo]: (Pick a point, enter x,y,z coordinates, or choose an option, or press Enter to exit.)

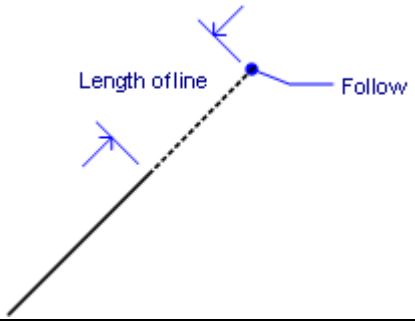
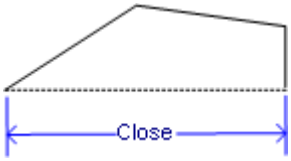
Set end point or [Angle/Length/Follow/Close/Undo]: (Pick a point, enter x,y,z coordinates, or choose an option, or press Enter to exit.)



Command Options

Option	Description
Start of line	Specifies the starting point of the first line segment.
Last point	Press Enter to use the most recently specified point. 
End point	Specifies the endpoint of each line segment.
Angle	Draws the line at a user-specified angle and distance; prompts you: Angle of line - specify the angle (measured counter-clockwise from the positive x-axis). Length of line - specify the length of the segment.

Length	Draws the line at a user-specified distance and angle; prompts you: Length of line - specify the length of the segment. Angle of line - specify the angle (measured counterclockwise from the positive x-axis).
Undo	Undoes (erases) the last line segment.
Follow	Draws the next line segment at the same angle as the previous segment. Length of line - specify the length of the segment. 
Close	Closes the "polygon" by drawing a line segment between the current endpoint and the first start point. 

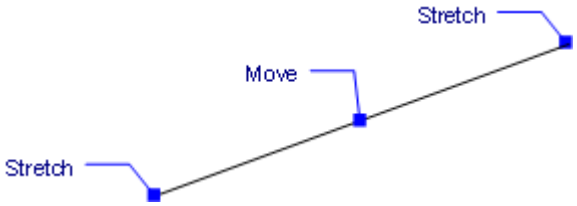
Procedures

Drawing lines

Grips Editing

Lines can be edited directly through grips:

1. Select the line. Notice that it has three grips.
2. Drag the grips to perform the following actions:
    - Drag an endpoint grip to stretch the line.
    - Drag the midpoint grip to move the line.


Stretch
Move
Stretch

Related Commands

MLine - draws multilines.
PLine - draws connected lines and arcs.
Ray - draws semi-infinite lines.
Sketch - draws sketched (freehand) lines.
Trace - draws wide line segments.
XLine - draws infinite lines.
Properties - changes the properties of lines through a dialog box.
Change - changes the properties of lines through the command bar.

Linetype

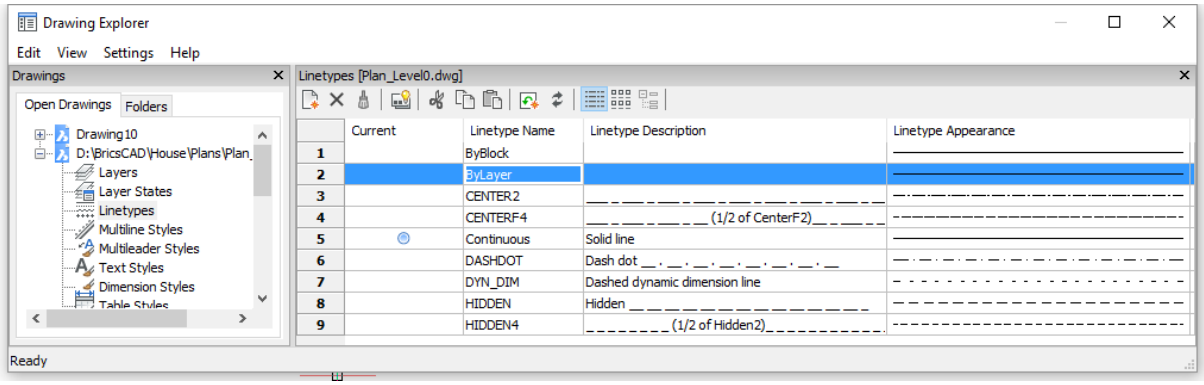
Loads, sets, and creates linetypes through the Drawing Explorer.

Accessing the Command

command bar: *linetype*
transparent: *'linetype*
aliases: *lt, ddltype, expltypes*
menu bar: *Tools | Drawing Explorer | Linetypes*
status bar: *Linetype* Field (*ByLayer*)

: ***linetype***

Displays the Linetypes node of the Drawing Explorer dialog box:



Add linetypes or modify their properties, and then click X.

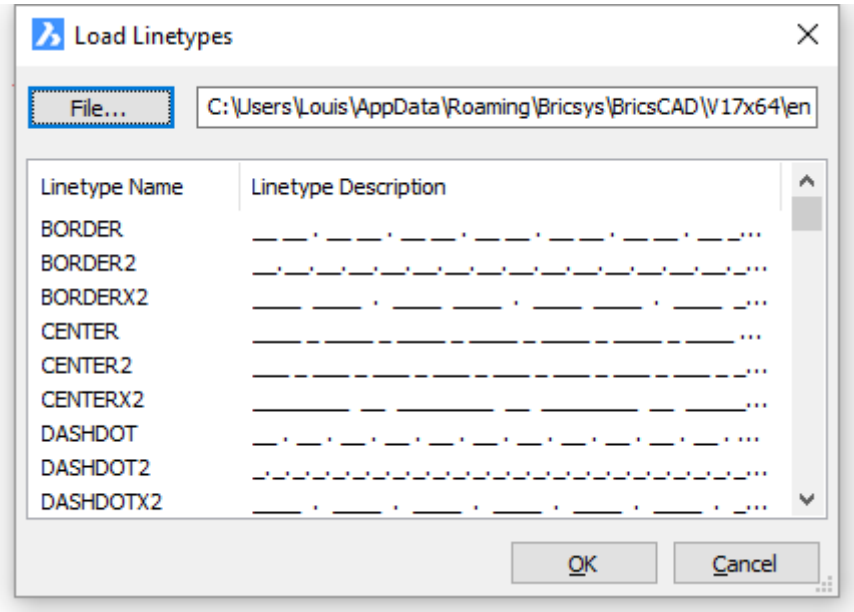
Press F1 to access help.

New drawings usually contain these three linetype definitions: Continuous, ByLayer, and ByBlock.

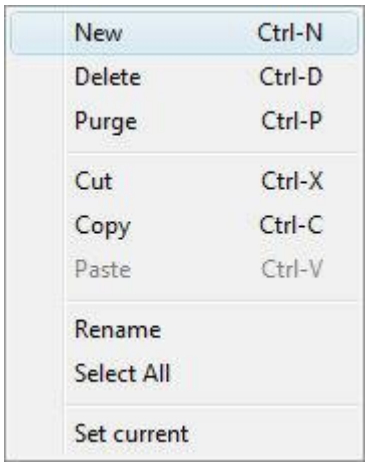
Any other linetype must be loaded into the drawing before it can be used: click the New button to load the linetype.

To start new drawings with all linetypes loaded, create and save the drawing as a DWT template file.

Command Options

Option	Description
New	Loads additional linetype definitions into the drawing. Displays the Load Linetypes dialog box:  (To add linetypes from a different LIN file, click the File button.) Select a linetype, and then click OK.
Delete	Erases linetype definitions from the drawing. The following linetype definitions cannot be deleted: • Continuous • ByLayer • ByBlock • Any linetype in use
Purge	Removes all unused linetype definitions from the drawing.

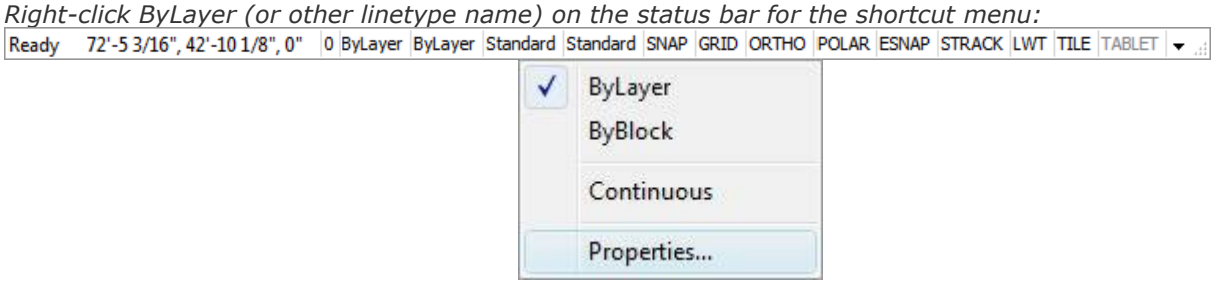
Shortcut Menu Options


New Ctrl-N
Delete Ctrl-D
Purge Ctrl-P
Cut Ctrl-X
Copy Ctrl-C
Paste Ctrl-V
Rename
Select All
Set current

Option	Keystroke	Description
New	Ctrl+N	Creates new linetype definitions.
Delete	Ctrl+D	Erases the selected linetypes.
Purge	Ctrl+P	Purges unused linetypes from the current drawing.
Cut	Ctrl+X	Cuts the selected linetype to the Clipboard.
Copy	Ctrl+C	Copies the selected linetype to the Clipboard.
Paste	Ctrl+V	Pastes the linetype definition into the current drawing.
Rename		Renames the selected linetype.
Select All		Selects all linetype definitions.
Set Current		Sets the selected linetype as current.

Status Bar Menu

Right-click ByLayer (or other linetype name) on the status bar for the shortcut menu:



Shortcut Menu	Description
ByLayer, ByBlock, Continuous	Chooses the default linetype. When the drawing contains additional linetypes, their names are listed here.
Properties	Opens the Linetypes section of the Drawing Explorer.

Related Commands

-Linetype - sets and loads linetypes through the command bar.
LWeight - specifies lineweights.
Settings - sets linetype parameters.
Layer - assigns linetypes to layers.
Properties - change linetypes through a palette.
Change and ChProp - change linetypes through the command bar.

Explorer - Opens the Drawing Explorer dialog box.

-Linetype

Loads, sets, and creates linetypes through prompts in the command bar.

Accessing the Command

command bar: *-linetype*
transparent: *'-linetype*
alias: *-lt*
status bar: *ByLayer*

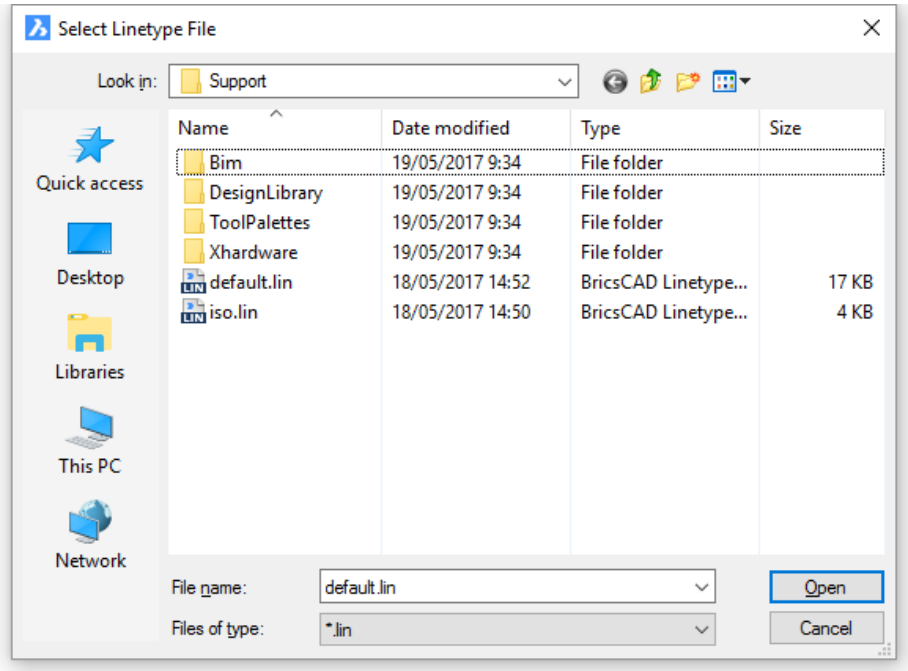
: ***-linetype***
Prompts you in the command bar:
Linetype [? to list/Create/Load/Set]: *(Enter an option.)*

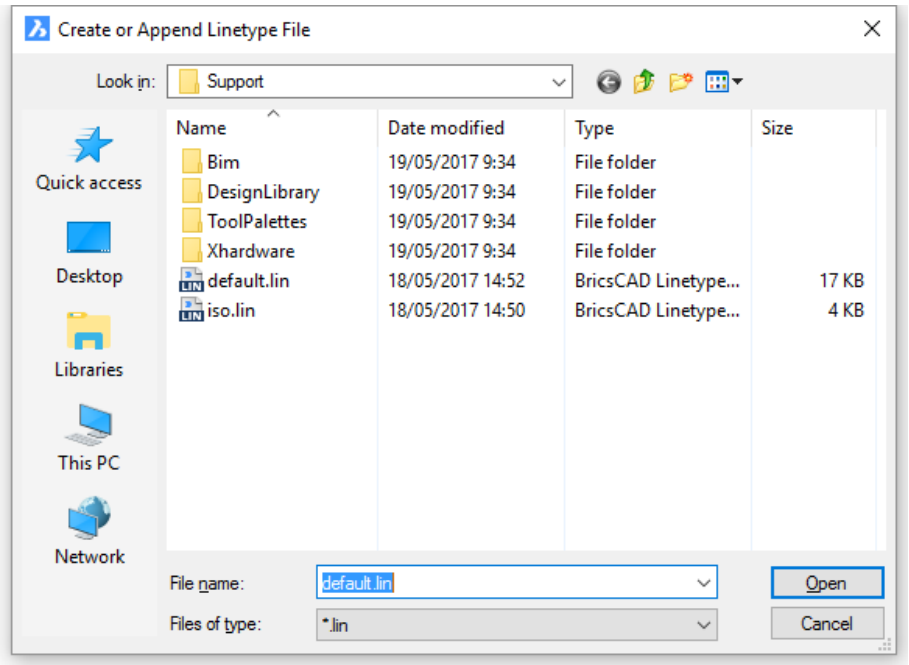
This command repeats itself until you press Enter or Esc.

New drawings usually contain just three linetype definitions: Continuous, ByLayer, and ByBlock. Any other linetype must be loaded into the drawing before it can be used, as follows:

1. Use the **-Linetype** command's Load option to load the linetype.
2. Use the Set option to set the linetype.

Command Options

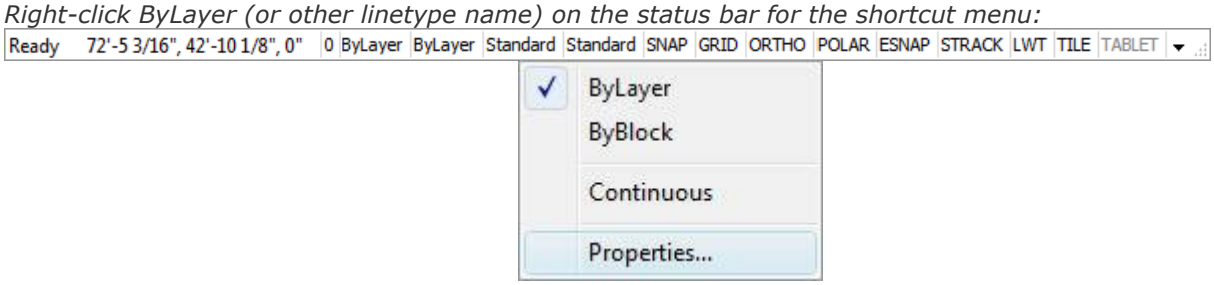
Option	Description
? to list	Lists the names of linetypes in a LIN file; displays the Select Linetype File dialog box:  Choose a LIN file, such as default.lin, and then click Open. BricsCAD lists the names and details of the file's linetype definitions in the text window.
Create	Creates new linetype definitions.

Name for new linetype	Specifies the name of the new linetype definition. After you enter a name, such as "new," BricsCAD displays the Create or Append Linetype File dialog box:  You add the new linetype definition to an existing LIN file: select its file name, and then click Open. BricsCAD checks if the name exists in the file: **One moment... Checking existing linetypes for "new."** And then carries on with the next prompt.
Linetype description	Describes the linetype, such as "Legal border."
Linetype definition	Defines the linetype pattern. Use the following coding: • Dashes - positive numbers, such as 0.5. • Spaces - negative numbers, such as 0.25. • Dots - zero (0). Separate numbers with commas, such as 0.5, -.25, 0. BricsCAD adds the definition to the LIN file, and reports: **Linetype "new" was defined in "C:\...default.lin"**
Load	Loads a linetype definition from a LIN file. Prompts you: Enter linetype to load - specify the name of the linetype, such as 'border'. Displays the Select Linetype File dialog box; choose a LIN file, and then click Open. BricsCAD reports, "Linetype 'BORDER' loaded."
Set	Sets a loaded linetype as the current (working) linetype.
? to list linetypes in drawing	Lists the linetype definitions in the drawing.

Linetype to set as current	Specifies the name of the linetype to set as the working linetype.

Status Bar Menu

Right-click ByLayer (or other linetype name) on the status bar for the shortcut menu:



Shortcut Menu	Description
ByLayer, ByBlock, Continuous	Chooses the default linetype. When the drawing contains additional linetypes, their names are listed here.
Properties	Opens the Linetypes section of the Drawing Explorer. See Linetype command.

Related Commands

Linetype - sets and loads linetypes through the Drawing Explorer.
LWeight - specifies lineweights.
Settings - sets linetype parameters.
Layer - assigns linetypes to layers.
Properties - change linetypes through a palette.
Change and ChProp - change linetypes through the command bar.

List

Lists the properties of selected entities.

Accessing the Command

command bar: *list*
aliases: *li, ls*
menu bar: *Tools | List Entities*
toolbar: *Inquiry |*

: ***list***

Prompts you in the command bar:

SOrt/Tracking/<Select entities to list>: *(Choose one or more entities, or enter an option.)*

To better see the data returned by this command, press F2 to switch to the text screen.
If the listing gets too long, press Esc to end it.

Command Options

Option	Description
SOrt	Switches from sequential to sort mode. When you select more one entity, this option lists them sorted by entity. For example, select a line, an arc, and another line. This option lists them as line 1, line 2, and arc 1.
Tracking	Specifies the number of lines to display at a time; prompts you: Number of command lines to track <256> - specify the number of lines to display before prompting you "Press ENTER to continue."
Select entities to list	Selects the entities to be listed.
SEquential	Switches from sort to sequential mode. When you select more than one entity, this option lists them by the order in which you selected them. For example, select a line, an arc, and another line. This option lists them as line, arc, and line.

A typical List listing looks like this:

```
------ Line 0 ----------------------------------------------------
                 Handle:  B9
          Current space:  Model
                  Layer:  0
                  Color:  256(BYLAYER)
              Linetype:  ByLayer
            From point:  X=   -36.0000  Y=   16.5000  Z=   0.0000
              To point:  X=   -24.5000  Y=   11.5000  Z=   0.0000
                 Length:  12.5399
     Angle in UCS XY plane:  337
           Delta values:  X=   11.5000  Y=   -5.0000  Z=   0.0000
```

Related Commands

Area - reports the area, circumference, and perimeter of entities.
DbList - reports parameters of every entity in the drawing.
Dist - reports the 2D and 3D distances between two points.
Id - reports the x,y,z coordinates of picked points.
MassProp - reports the properties of solid models and regions.
Properties - lists the area and other parameters of entities in a palette.

LiveSection

Toggles the *Live Section* property of a section plane.

Accessing the Command

command bar: *livesection*
menu bar: *Model* | *Sections* | *Livesection*
toolbar: *Sections* |

: ***livesection***
Prompts you in the command bar:
Select a section entity: *(click a section entity in the drawing)*
If the *Live Section* property of a section plane was off, it is turned on and vice versa.

Command Options

No options.

Related Commands

ClipDisplay - toggles the Clip Display property of a section plane or a BIM section entity.
BimSection - creates a BIM section entity.
BimSectionOpen - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.
Section - creates section planes of 3D solids; the results are region entities.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.
SectionPlaneToBlock - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

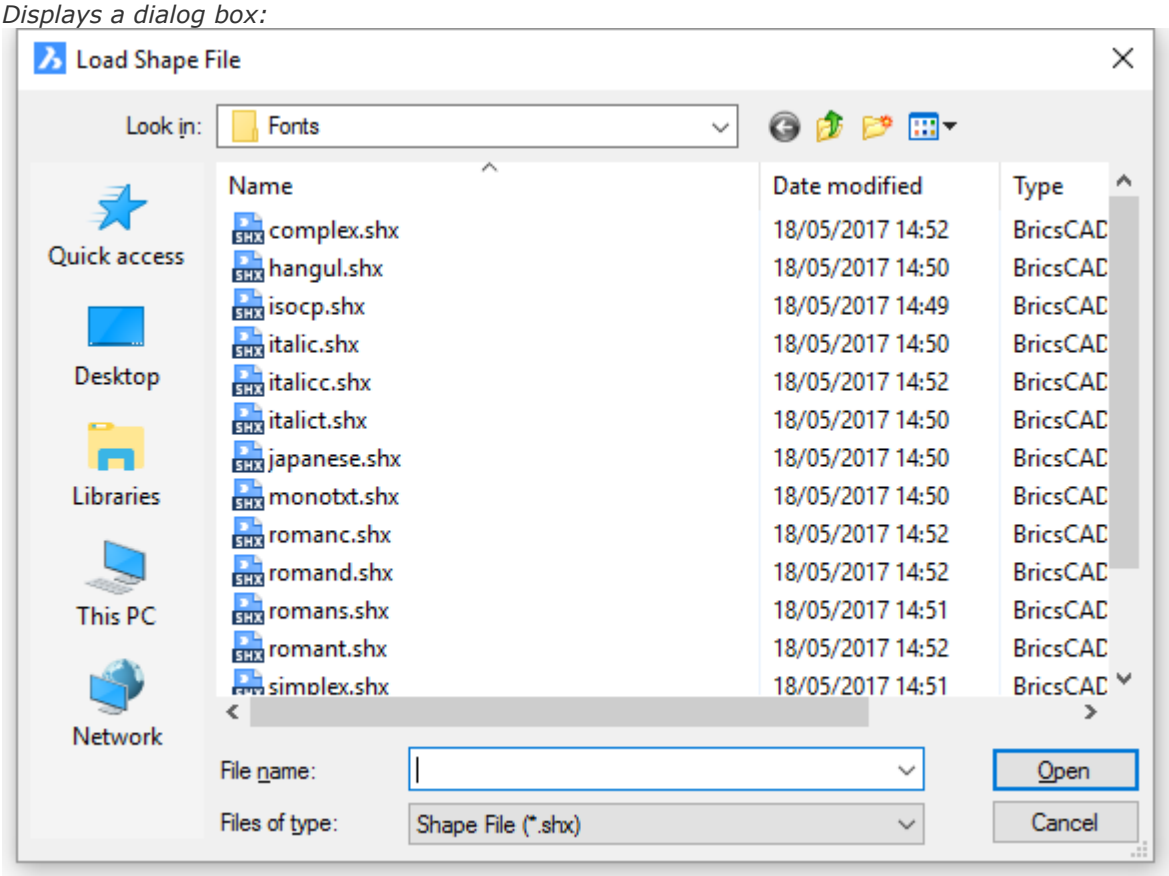
Load

Loads SHX compiled shape files into the current drawing.

Accessing the Command

command bar: *load*

: ***load***
Displays a dialog box:



Choose an SHX file, and then click **Open**.

SHX files are compiled SHP shape files, which are used to define fonts and shapes. This command works only with shape SHX files, not font SHX files. Shapes were an early version of blocks. They were highly efficient but difficult to create. This command remains in BricsCAD largely for compatibility reasons.

Shapes are placed with the **Shape** command.

Command Options

No options.

Related Commands

Shape - inserts shapes into drawings.
Insert - inserts blocks into drawings.

Loft

Creates 3D solids passing through two or more cross sections.

Accessing the Command

command bar: *loft*
menu bar: *Model | 3D Solids | Loft*
toolbar: *3D Modeling |*

: ***loft***
Prompts you in the command bar:

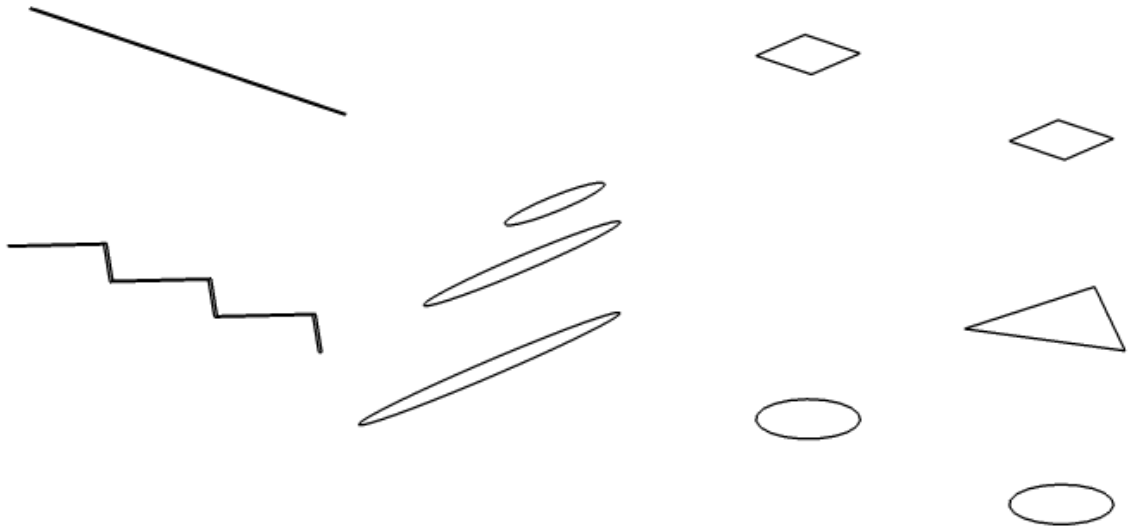
Select cross sections in lofting order [MOde]: (Select a closed 2D entity or a planar face of a 3D solid; or choose the Mode option.)

Select cross sections in lofting order [MOde]: (Select a closed 2D entity or a planar face of a 3D solid; or choose the Mode option.)

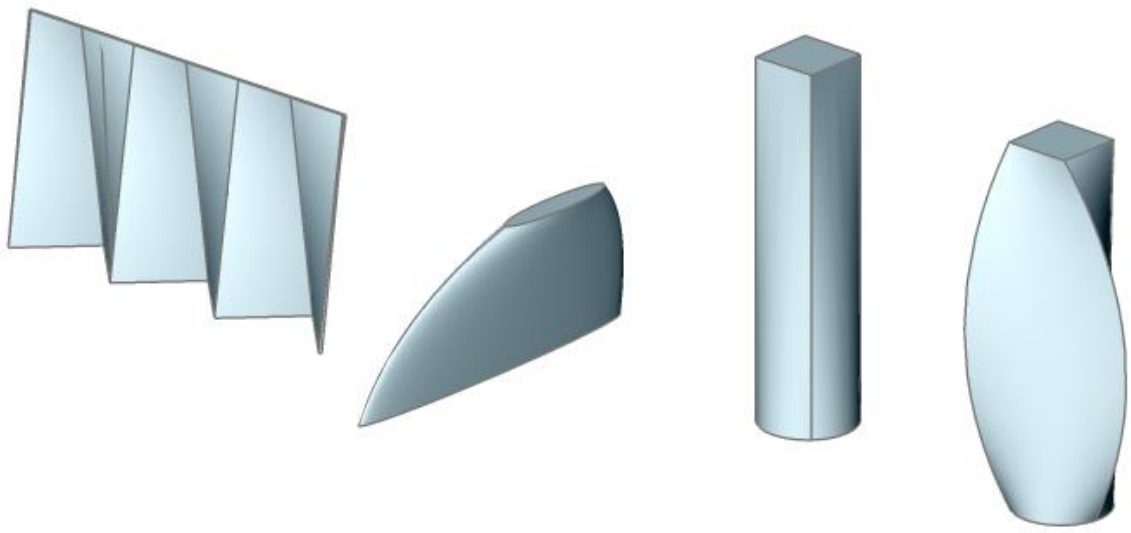
The first part of the lofted 3D solid is created.

Select cross sections in lofting order [MOde]: (Select more 2D entities or planar faces; or choose the Mode option; or press Enter to stop selecting cross sections.)

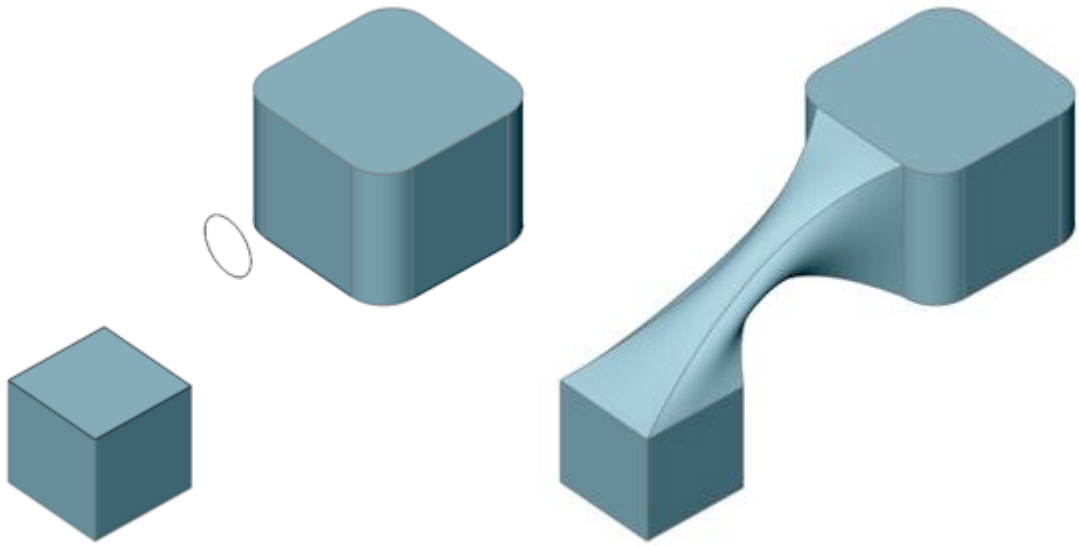
The the lofted 3D solid is created.
3 cross sections selected
Enter an option [Guides/Cross sections only/Settings] <Cross sections only>: *(Press Enter or choose an option.)*



Cross sections

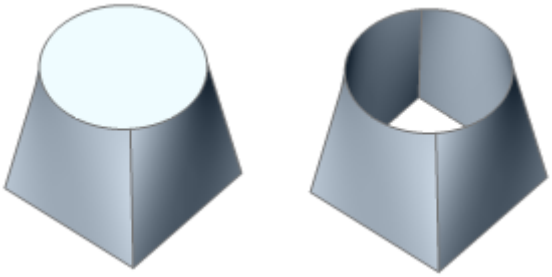


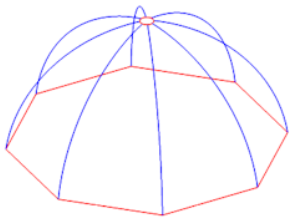
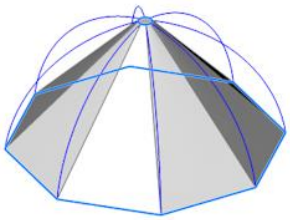
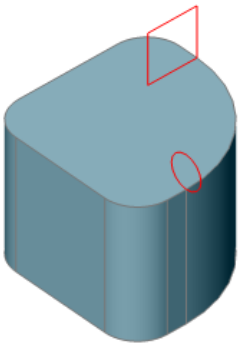
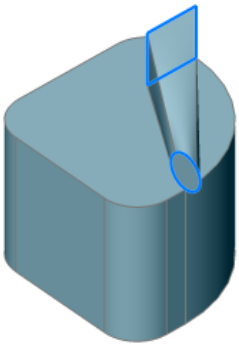
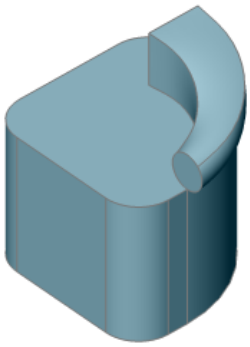
Lofted solids.



Lofted solid using faces of 3D solids and a circle.

Command Options

Option	Description
Mode	Allows to create a solid (left) or a surface (right).  Prompts you: **Choose type of created entity [SOlid/SUrface] <SOlid>:** • **Solid**: Creates a lofted solid. • **Surface**: Creates a lofted surface.
Cross Sections only	Uses the selected cross sections only, no guides. If DELOBJ = 1 or 2, the selected 2D entities are deleted.
Guides	Use guide lines between the selected cross sections. If DELOBJ = 2, the selected guide entities are deleted. Prompts you: **Select guide curves:** Select a linear entity (lines, arcs, elliptical arcs, splines, single-segment 2D polylines and edges of 3D solids are accepted.) **Select guide curves:** Select more linear entities or press Enter.

		
Cross sections (red) and guides (blues)	*Cross sections only*	*All guides selected*
		

	Cross sections (red).	*Cross sections selected only*	*Curved solid edge selected as a guide.*

Settings

Allows to set the Loft surfaces and solids settings.
The default settings are controlled through a series of system variables, which are saved in the drawing:

Lofted surfaces and solids	
Loft angle 1	90.0000
Loft angle 2	90.0000
Loft magnitude 1	0
Loft magnitude 2	0
Loft normals	[1] Smooth surface
Loft param	0x0007 (7)
1	☑ No twist between cross sections
2	☑ Align directions of cross sections
4	☑ Create simple surfaces and solids
8	☐ Close between the starting and ending cross sections

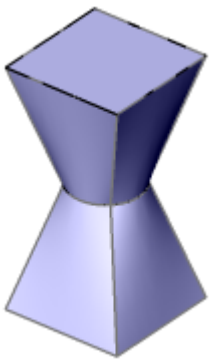
Loft Normals settings:

[0] Ruled surface
[1] Smooth surface
[2] Surface is normal to the first cross section
[3] Surface is normal to the last cross section
[4] Surface is normal to the first and last cross sections
[5] Surface is normal to all cross sections
[6] Surface uses draft angle and magnitude

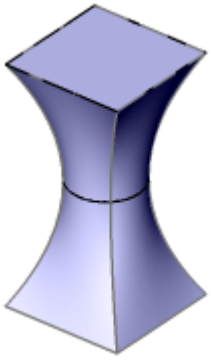
Prompts you:

Current option is Smooth. Enter an option [Ruled/Smooth fit/Normal to/Draft angles] <eXit>:

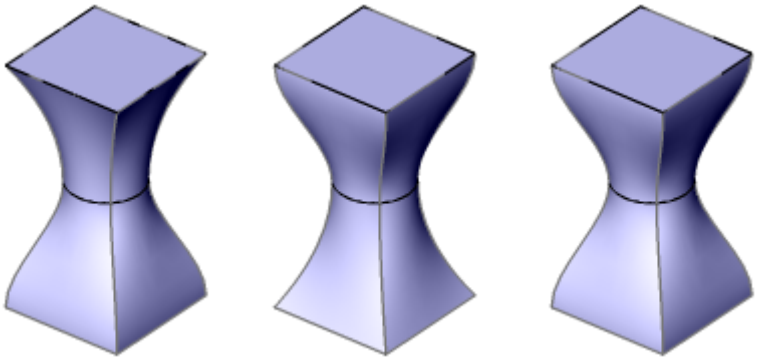
- **Ruled**: The surface of the solid is straight between the cross sections and has sharp edges at each cross section.



- **Smooth fit** (default): The surface of the solid is drawn smoothly between the cross sections.

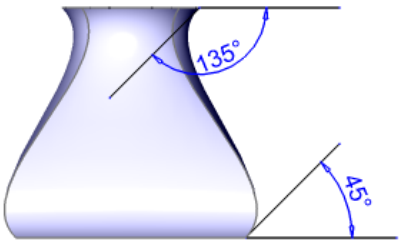


- **Normal to**: Allows to specify that the surface of the solid is normal to the cross sections. The options are **Start**, **End**, **Both** (= Start and End) and **All**.



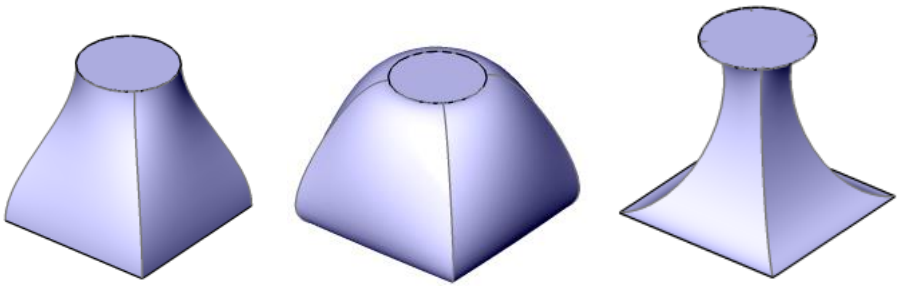
Solid surface is normal to the start (left), end (middle), both (right) cross sections

- **Draft Angles**: The draft angle is the start/end direction of the surface of the lofted solid.
  The **Magnitude** setting controls the relative distance of the surface from the cross section in the direction of the draft angle before the surface starts to bend toward the next cross section.



LOFTANG1 = 45°, LOFTANG2 = 135°

2



Loft Angles = 90° (left), Loft Angles = 0° (middle), Loft Angles = 180° (right)

Related Commands

Extrude - extrudes 2D entities to 3D solids.
dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmRevolve - creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmTwist - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.
Revolve - revolves 2D closed entities to turn them into 3D solids.
RevSurf - creates a 3D mesh surface by revolving a linear entity around a line.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.

LogFileOff

Turns off log file recording. Sets the value of the LOGFILEMODE system variable OFF.

Accessing the Command

command bar: *logfileoff*

: ***logfileoff***

No prompts at the command bar.

You can open the log file in Notepad or another text editor. The file is found in the folder specified by the LogFilePath system variable.

The default name of the file is drawingname_year-month-date_hour-minutes-seconds.log, such as drawing1_2009-08-31_08-32-46.log.

Command Options

No options.

See the Log Files section of the **Settings** dialog box.

Related Commands

LogFileOn - turns on log file recording.
Settings - sets up log file parameters in the Log Files section of the Settings dialog box.
CopyHist - copies command history from the Text window.

LogFileOn

Turns on log file recording; records all command prompts to a LOG file. Sets the value of the LOGFILEMODE system variable ON.

Accessing the Command

command bar: *logfileon*

: ***logfileon***

No prompts at the command bar.

BricsCAD records all command text to the LOG file located in the folder specified by the LogFilePath system variable.

The default name of the file is drawingname_year-month-date_hour-minutes-seconds.log, such as drawing1_2009-08-31_08-32-46.log.

Command Options

No options.

See the Log Files section of the **Settings** dialog box.

Related Commands

LogFileOff - turns off log file recording.
Settings - sets up log file parameters in the Log Files section of the Settings dialog box.

LookFrom

Controls the display of the Look From control and allows access to the related settings.

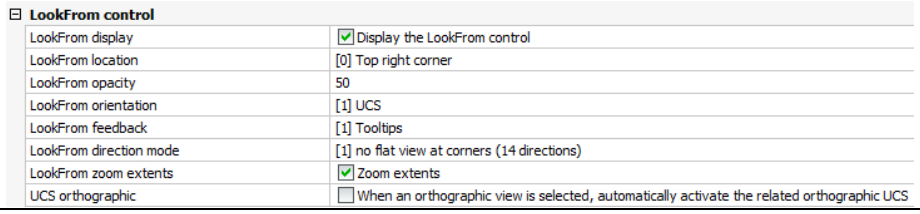
Accessing the Command

command bar: *lookfrom*

: ***lookfrom***

Prompts you in the command bar:
LookFrom [ON/OFf/Settings] <ON>: *(Choose an option.)*

Command Options

Option	Description
ON	Switches the display of the Look From control on: Move the cursor over the Look From control to activate it: The Look From control widget displays at the location which is defined by the NAVVIEWCUBELOCATION system variable. The default location is the top right corner of the drawing area.
OFF	Switches the display of the Look From control off.
Settings	Displays the Look From control settings in the Settings dialog box.  **LookFrom control** LookFrom display: Display the LookFrom control LookFrom location: [0] Top right corner LookFrom opacity: 50 LookFrom orientation: [1] UCS LookFrom feedback: [1] Tooltips LookFrom direction mode: [1] no flat view at corners (14 directions) LookFrom zoom extents: Zoom extents UCS orthographic: When an orthographic view is selected, automatically activate the related orthographic UCS

Procedures

Setting the Look From Control system variables and user preferences
Setting the Look From mode
Using the Look From control in Isometric mode
Using the Look From control in Twist mode

Related Commands

No related commands

LWeight

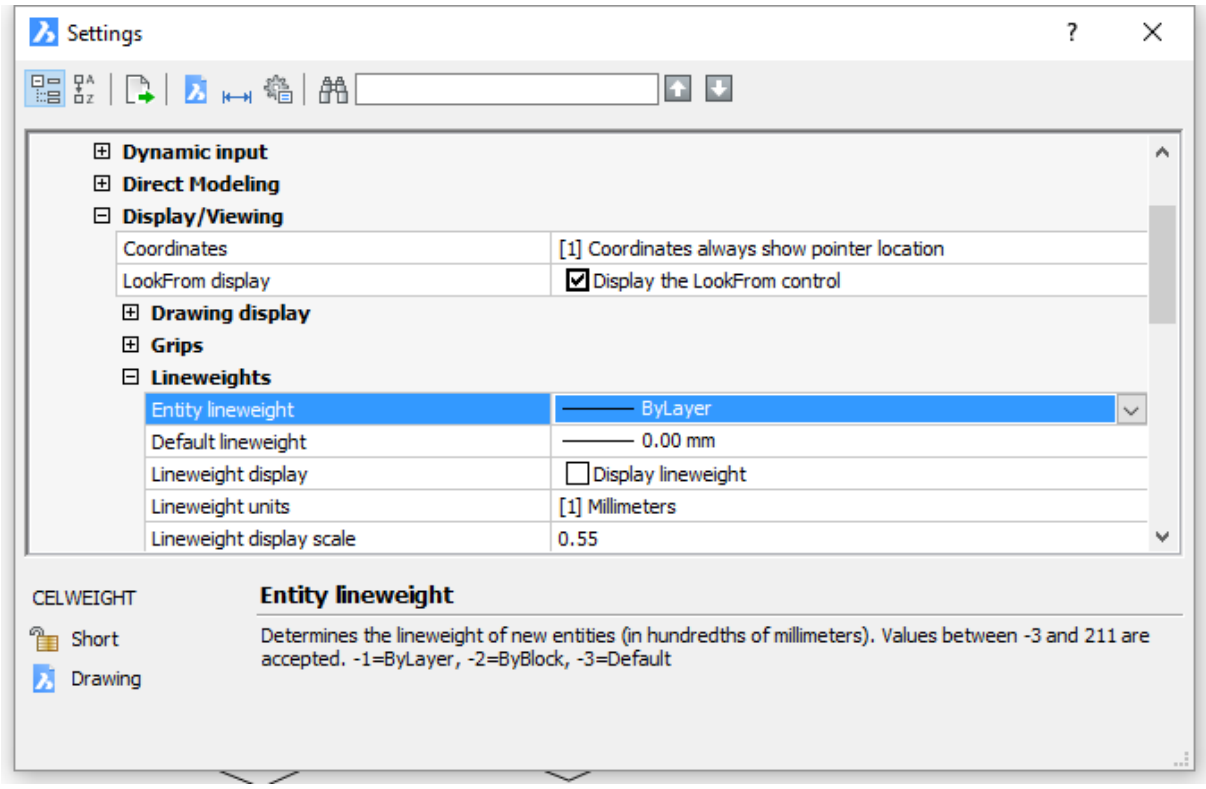
Controls the display of lineweights in the drawing through the Settings dialog box.

Accessing the Command

command bar: *lweight*

toolbar: *Entity Properties* | ByLayer
status bar: *LWT*

: ***lweight***
Displays the Lineweights section of the Settings dialog box:

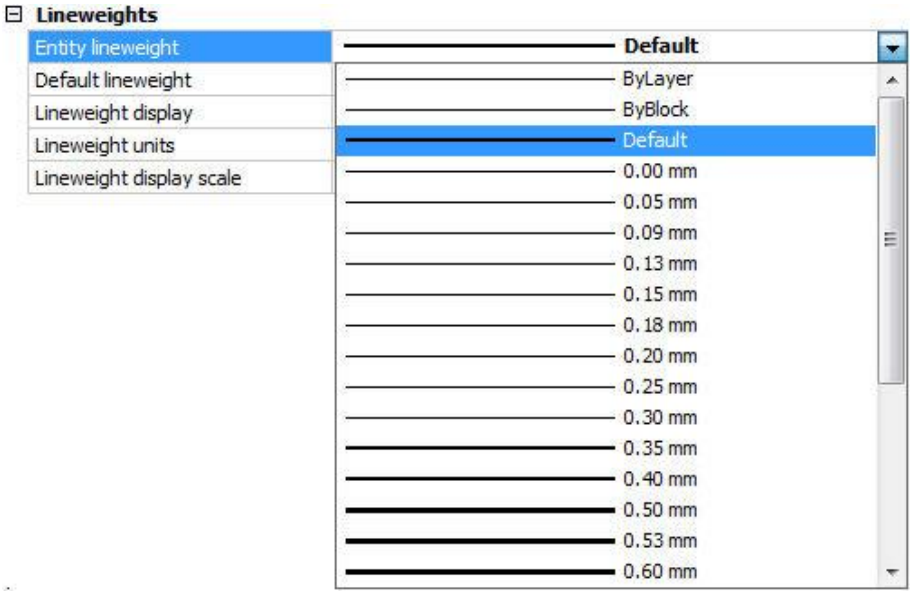


*Change settings, and then click **X**.*

Lineweights can also be specified by the following commands:

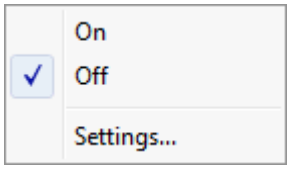
- The **Layer** command.
- The droplist in the Entity Properties toolbar.
- The droplist in the **Properties** palette.

Command Options

Option	Description
Entity Lineweight	Specifies the lineweight for entities drawn from now on; choose a preset lineweight from the droplist.  This sets the current lineweight in the Entity Properties droplist and Properties palette. Lineweights cannot be customized, unlike linetypes.
Default Lineweight	Specifies the default lineweight for all new drawings; choose a preset lineweight from the droplist.
Lineweight Display	Toggles the display of lineweights: • On - displays lineweights. • Off - does not display lineweights; all entities have the same lineweight.
Lineweight Units	Specifies the unit of measurement: • Millimeters - measures the width in millimeters. • Inches - measures the width in inches.
Lineweight Display Scale	Specifies the relative scale of lineweights; applies to lineweights in Model tab only.

Status Bar Menu

Right-click LWT on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on lineweights.
Off	Turns off lineweights.
Settings	Opens the Lineweights section of the Settings dialog box. ⊟ **Drawing** ⊟ **Drafting** ⊞ **Drawing units** ⊞ **Coordinate input** ⊟ **Display/Viewing** ⊞ **Drawing display** ⊞ **Grips** ⊟ **Lineweights** Entity lineweight — ByLayer Default lineweight — 0.50 mm Lineweight display — ☑ Display lineweight Lineweight units — Millimeters Lineweight display scale — 0.25

Related Commands

Settings - sets lineweight parameters.
Layer - assigns lineweights to layers.
Properties - change lineweight through a palette.
Change and ChProp - change lineweight through the command bar.
Plot and Print - optionally plot entities with lineweights.

M

Mail

Attaches the current drawing to a new message with your computer's default email client.

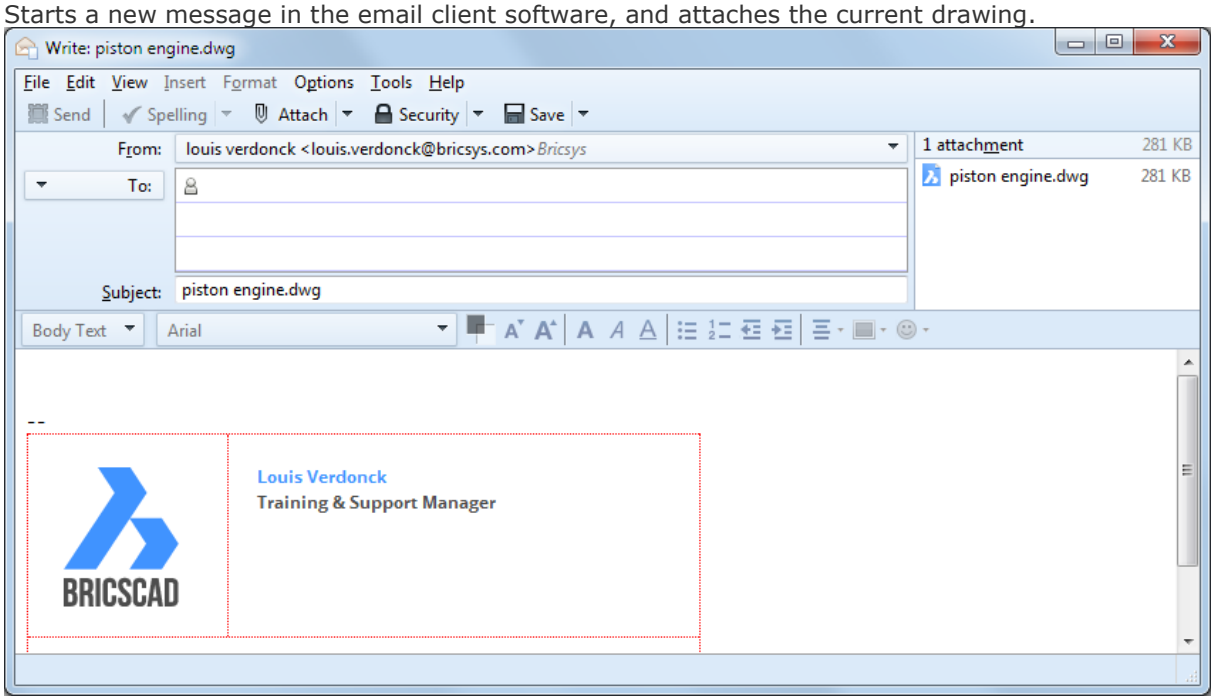
Accessing the Command

command bar: *mail*
menu bar: *File | Send Mail*

: ***mail***

No prompt in the command bar.

Starts a new message in the email client software, and attaches the current drawing.



*Complete the message, and then click **Send**.*

Command Options

No options.

Manipulate

Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl-key pressed while starting an action to manipulate a copy of the selected entities.

Accessing the Command

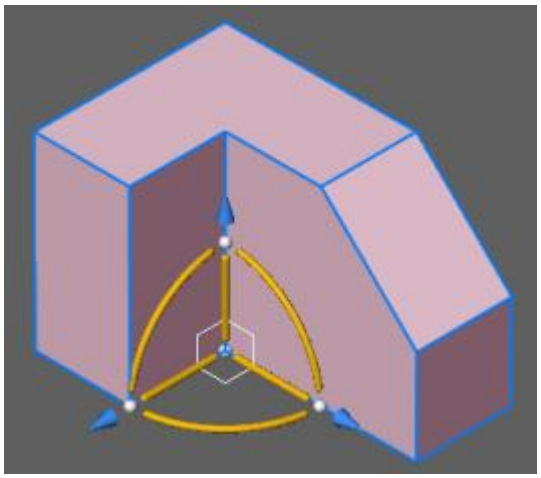
command bar: *manipulate*
Quad: *Modify | Manipulate*

: ***manipulate***
Prompts you in the command bar:

Select entities: (Select one or more entities.)
Entities in set: 1

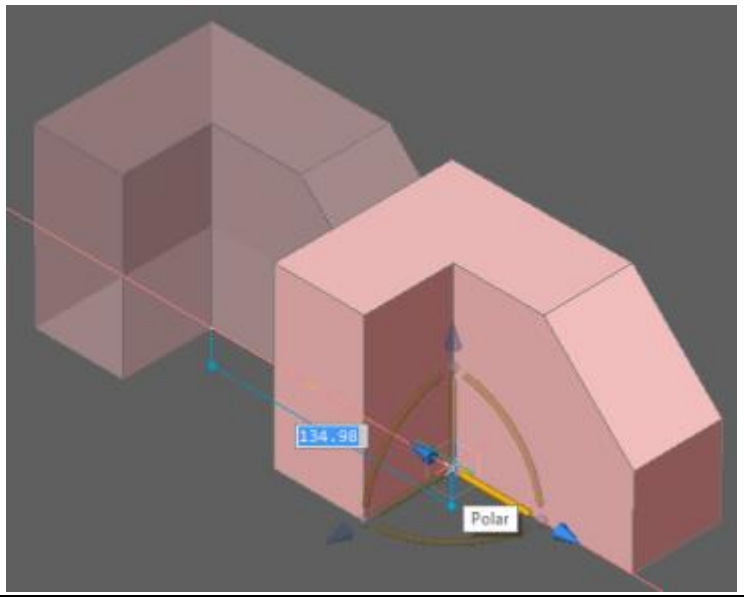
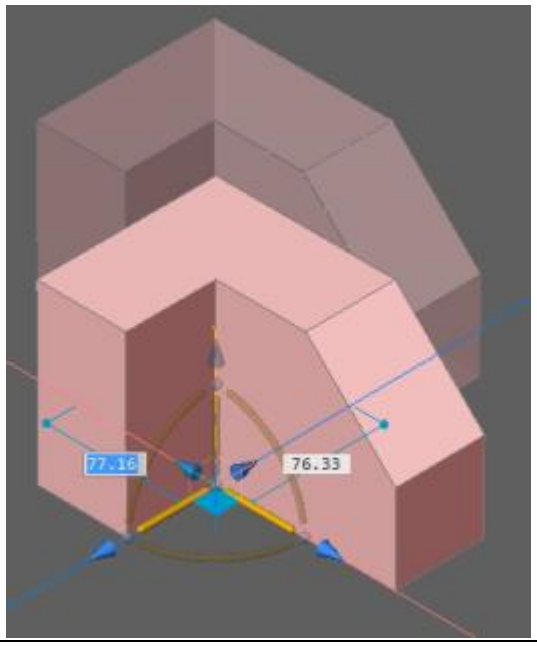
Select entities: (Select more entities or press Enter.)

The manipulator widget displays.

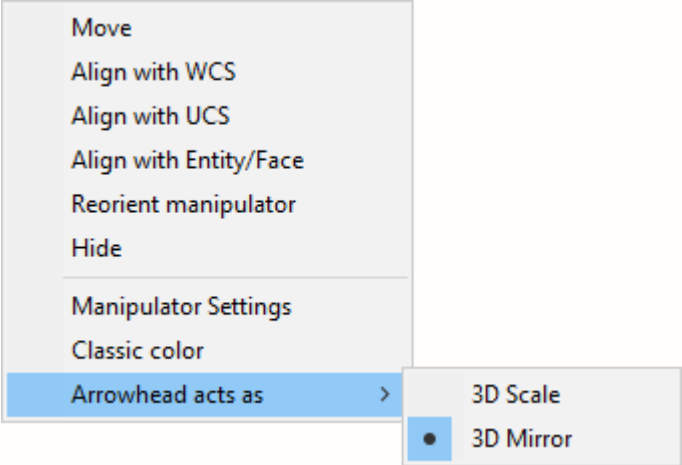


Command Options

Option	Description
Drag an axis	Moves or copies* the selection along the axis being dragged. Do the following: • Place the cursor over an axis and click. The selection moves along the axis with the cursor. • Pick a point or, if Dynamic Input is on, type a distance in the dynamic entry field.

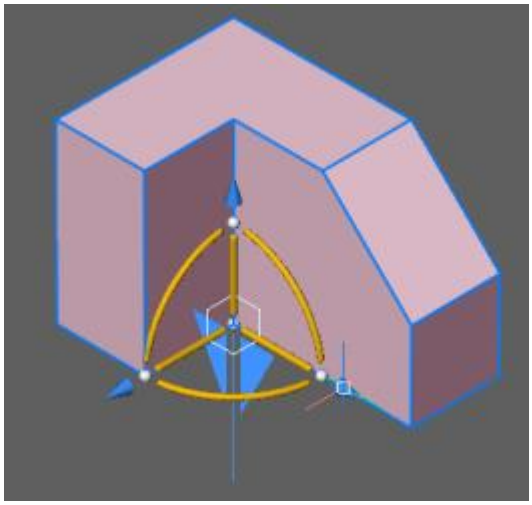
	
Drag a plane	Moves or copies* the selection in the plane being dragged. Do the following: • Place the cursor over one of the plane swatches. The swatch under the cursor highlights. • Click when the swatch of the desired plane highlights. The selection moves along with the cursor in the selected plane. • Pick a point or, if Dynamic Input is on, type a distance in the dynamic entry fields. Hit the TAB-key to switch between the entry fields; press Enter to confirm the current values. 
Drag an arrowhead	Mirrors, moves or scales the selection. Right click when the cursor is over the manipulator widget and right click to display a context menu.

Choose *Arrowhead act as*, then select the desired action: 3D Scale or 3D Mirror.

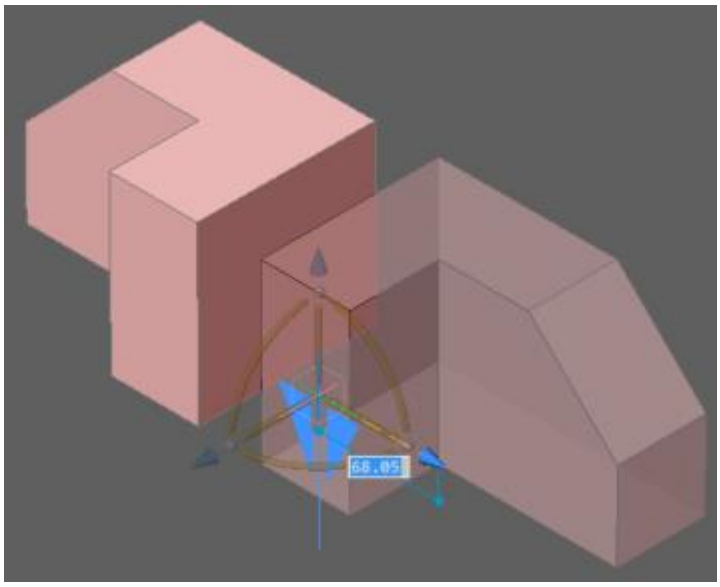


To mirror* the selection:

- Place the cursor over an arrowhead.
  The Mirror swatch displays in a plane through the selected axis and perpendicular to the mirror plane.
  The selection will be mirrored about the plane that is perpendicular to the axis of the selected arrowhead.

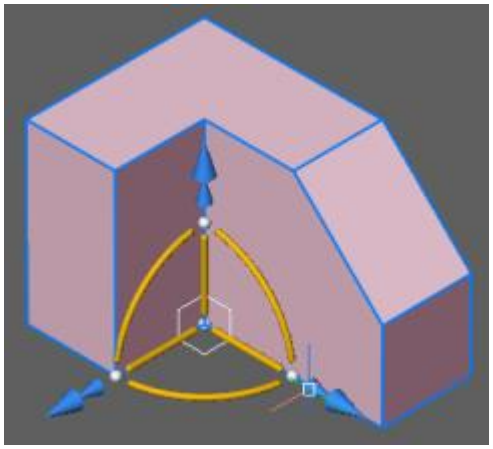


- Click when the desired mirror plane is selected.
  The selection is mirrored dynamically when you move the cursor.
- Pick a point, or if Dynamic Input is on, type a distance in the dynamic entry field.

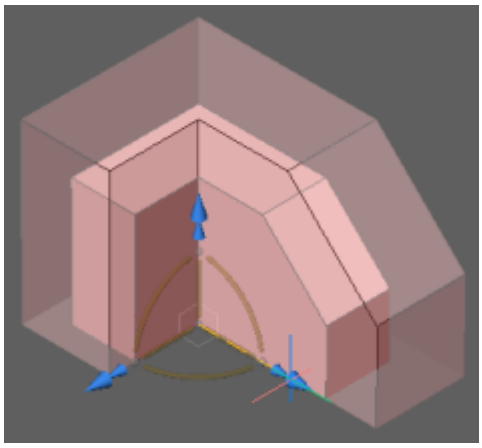


To scale* the selection:

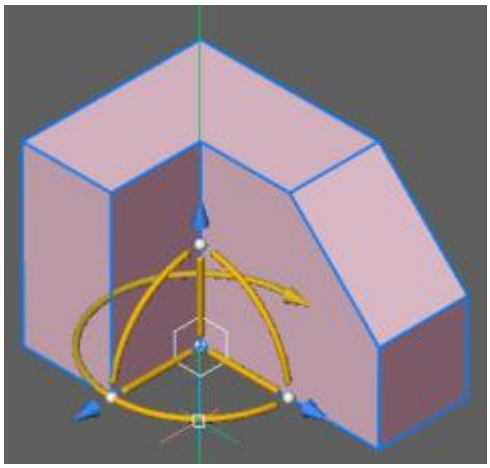
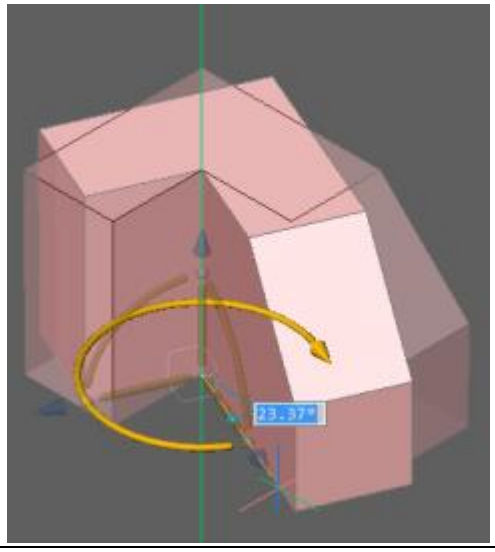
- Place the cursor over an arrowhead.
  The arrowheads are doubled with a large arrowhead.



- Click to start scaling the selection; you are prompted in the command bar:



*** MANIP_SCALE ***

	New scale factor [Reference]: Do one of the following: • Pick a point. • Type a scale factor at the command prompt. • Type R; prompts you: Enter reference length - use entity snaps to pick two points in the model. New length [Reference] - type the desired distance between the picked points.
Drag a rotation arc	Rotates the selection about one of the manipulator axes. Press and hold the Ctrl-key when starting the action to keep the original entities, thus creating a copy. Do the following: • Place the cursor over one of the three rotation arcs. The rotation arc under the cursor expands and the rotation axis displays.  • Click when the desired rotation axis displays. The selection rotates dynamically when you move the cursor. • Pick a point or, if Dynamic Input is on, type a distance in the dynamic entry field. 

Procedures

To define the manipulator settings
The manipulator layout
To control the location and orientation of the manipulator

Related Commands

Copy - copies entities.
dmMove - moves solids, faces or edges of a solid, or inserts using a vector.
dmRotate - rotates faces of a solid around an axis.
Move - moves entities.
Rotate - rotates entities in 2D.
Rotate3D - rotates entities about an axis in 3D space.
Scale - resizes entities.

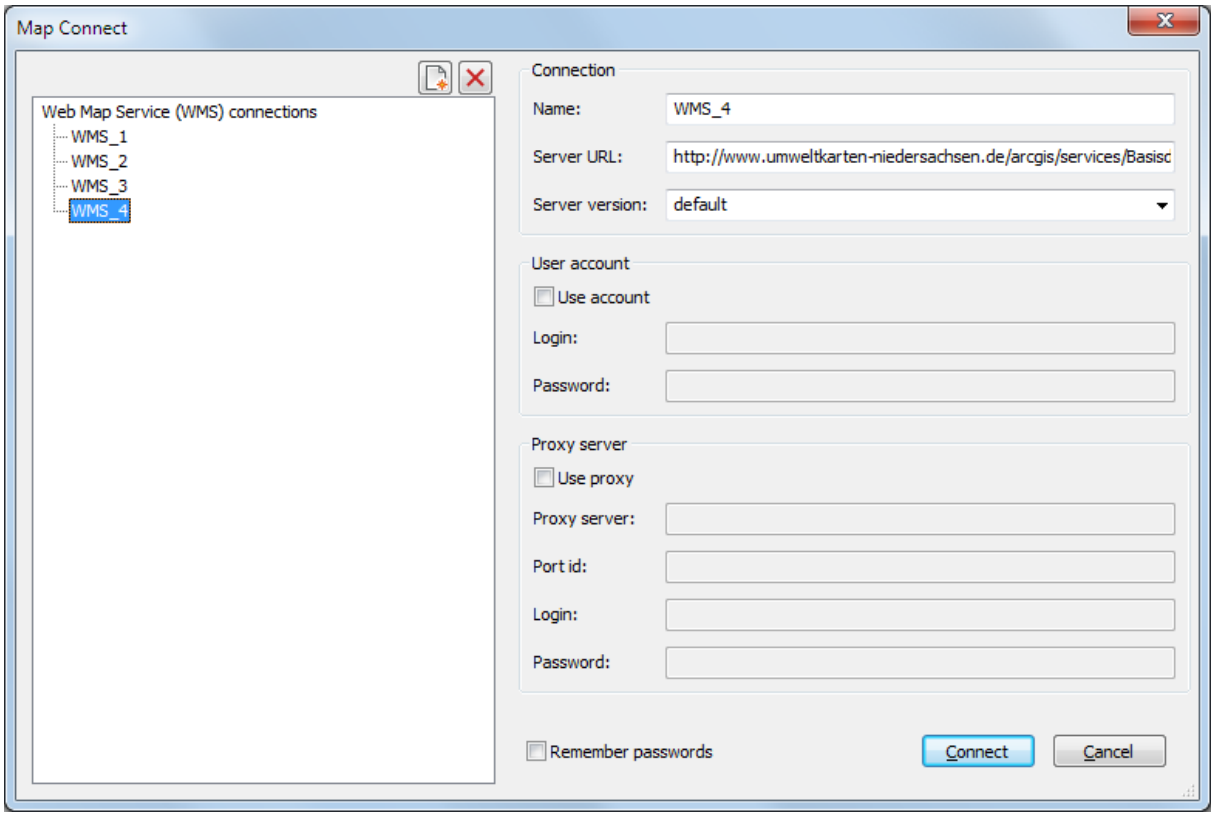
MapConnect

Allows to set up a connection with a Web Map Service through a dialog box.
Before adding map data to a drawing, the GEOGRAPHICLOCATION command must be executed to define and store the appropriate geographic location data in the drawing.

Accessing the Command

command bar: *mapconnect*

: ***mapconnect***
Displays a dialog box:



Command Options

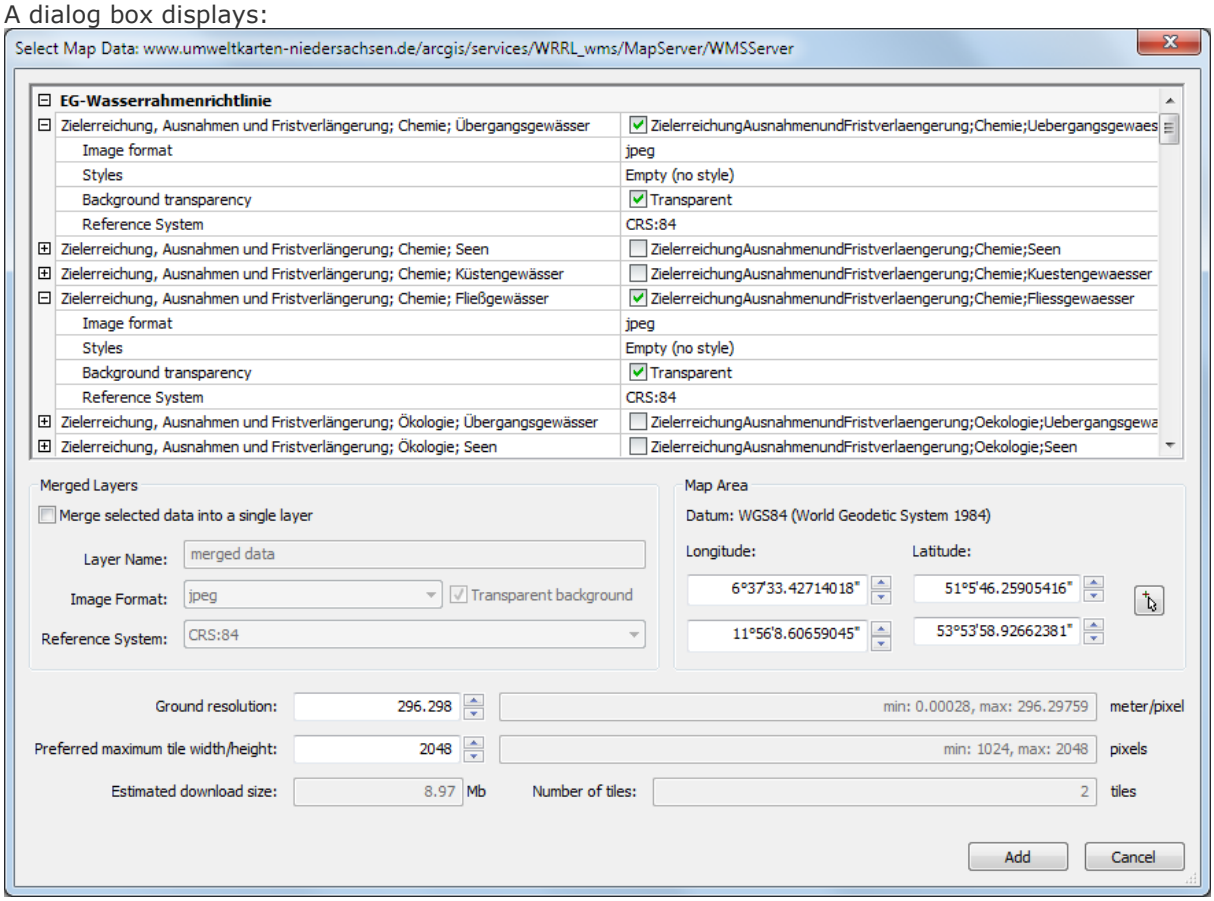
Option	Description
Name	(option) Type a name to replace the default WMS<x> name. The 10 most recently used connections are saved.
Server URL	Specify the URL of a Web Map Service.
Server version	Specify a server version or select one in the drop-down list.
User Account	
Use Account	Check the option if the Web Map Service uses a login and password.
Login	Type the login.

Password	Type the password
Proxy Server	
Use Proxy	Check the option if you want to use a proxy server to connect to the Web Map Server.
Proxy server	Type the URL of the proxy server.
Port Id	Specify the port to use for the connection.
Login	Type the login.
Password	Type the password.
Remember passwords	Check the option if you want passwords to be remembered for future use.

Do one of the following:

- Select a previously used Web Map Service in the list.
- Fill in the necessary fields and click the *Connect* button.

A dialog box displays:



Do the following:

- Select one or more data layers
- Specify the desired ground resolution and the preferred maximum dimensions of a tile.
- An estimate of the download size displays in the *Estimated download size* field.
- Press the *Add* button.

Downloaded map tiles are stored as regular image files in the folder where the drawing is saved.

The definition of the supported Coordinate Reference Systems (CRS) is stored in the new *geodatabase.xml* file which is located in the folder pointed to by the ROAMABLEROOTPREFIX variable, e.g. *C:\Users\John\AppData\Roaming\Bricsys\BricsCAD\V15x64\en_US\Support*

Also contained in *geodatabase.xml* is a list of cities and their WGS84 geo coordinates which is used by the "Choose Location..." functionality of the Geographic Location dialog.

Related Commands

Geographiclocation - sets the geographic location of the drawing.
Image - inserts images in drawings through the Drawing Explorer.

MassProp

Reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

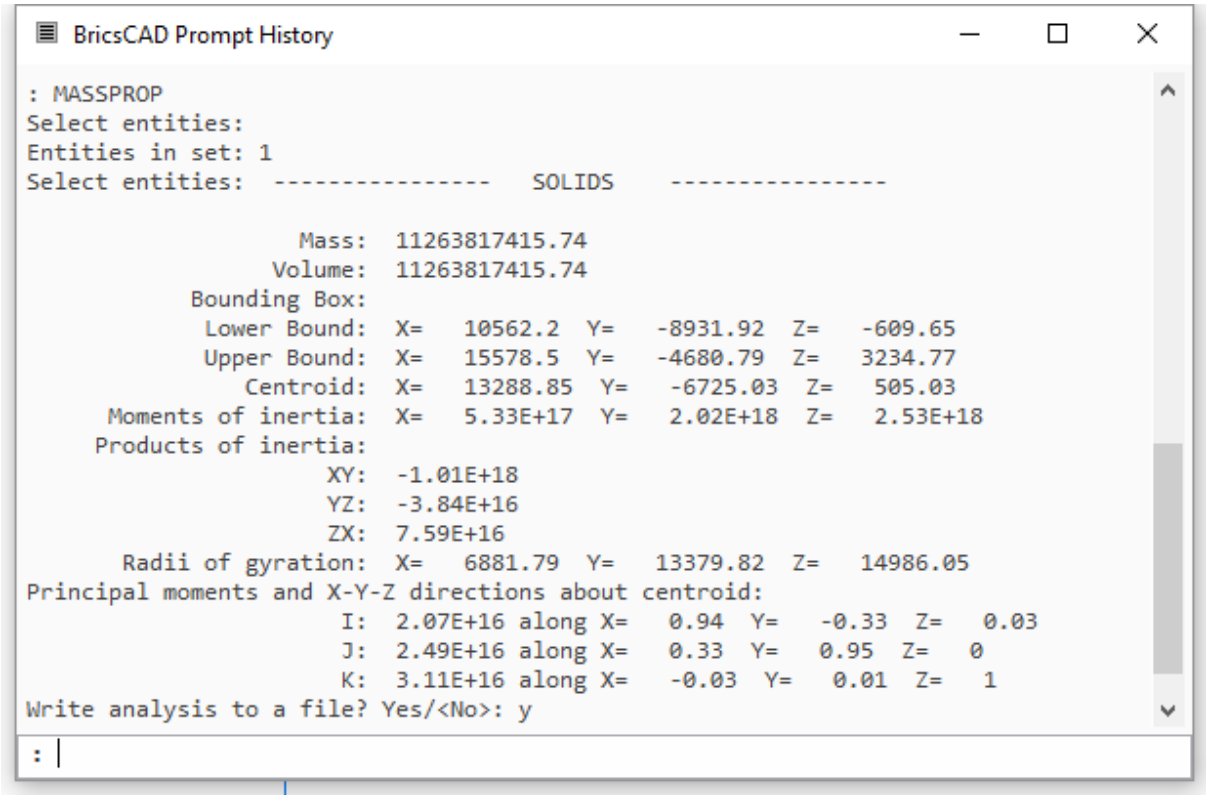
Accessing the Command

command bar: *massprop*
menu bar: *Tools | Inquiry | Mass Properties*
toolbar: *Inquiry |*

: ***massprop***
Prompts you in the command bar:
Select entities: *(Choose one or more ACIS objects.)*
Entities in set: 1

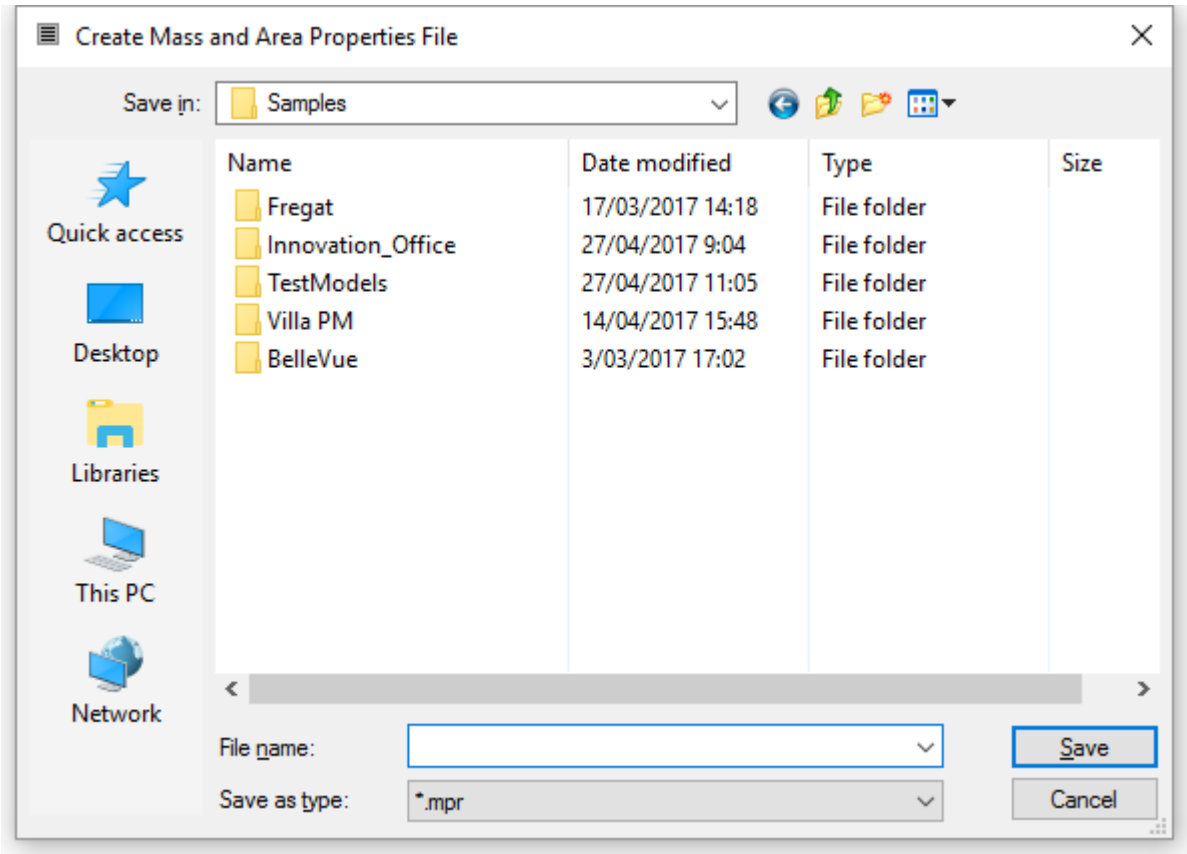
Select entities: (Choose one or more ACIS objects.)

Displays report in the command bar; press F2 to view report in the prompt history window:



Write analysis to a file? (Type Y or N.)

When you enter Y, displays dialog box:



Name the file, and then click Save.

The report is stored in an MPR file, which can be opened in any text editor.
MPR is short for "mass properties report."

Command Options

Option	Description
Select entities	Specifies the entities on which to report properties. Choose the 3D solid models and 2D region entities; all other entities are ignored by BricsCAD.
Write analysis to a file?	Writes the report to a text file. Type Y or N: • Yes - saves the report to a file; opens the Mass and Area Properties File dialog box. Enter a file name, and then click Save. The report is stored in an MPR file, which can be opened in any text editor. • No - does not save the report to a file.

Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Region - creates 2D regions from 2D entities.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

MatBrowserClose

Closes the materials browser.

Accessing the Command

command bar: *matbrowserclose*

: ***matbrowserclose***

Command Options

No options.

Related Commands

MatBrowserOpen - opens the materials browser, which shows an overview of available high-resolution render materials.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

MatBrowserOpen

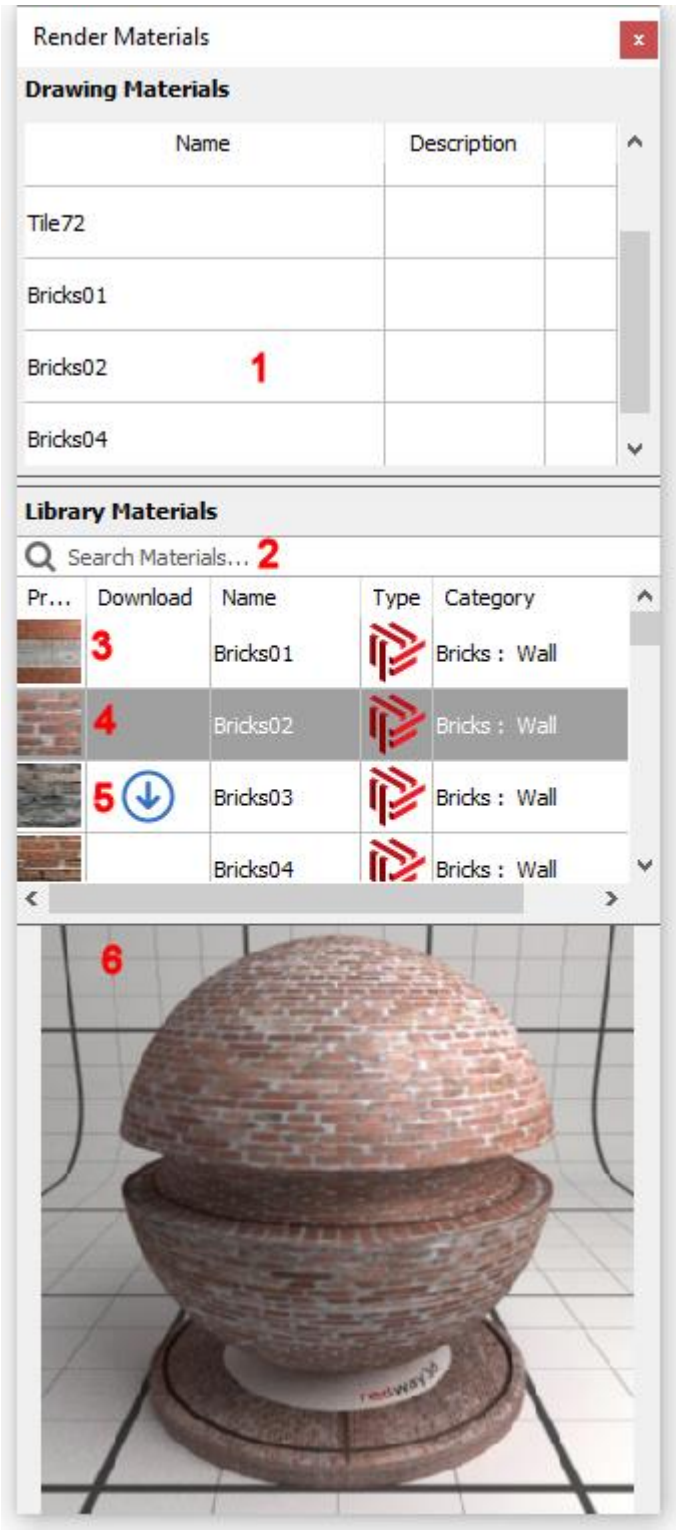
Opens the materials browser, which shows an overview of available high-resolution render materials.

Accessing the Command

command bar: *matbrowseropen*
aliases: *matb, matlib*

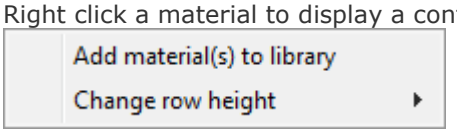
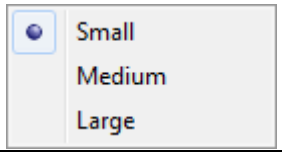
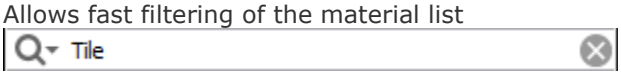
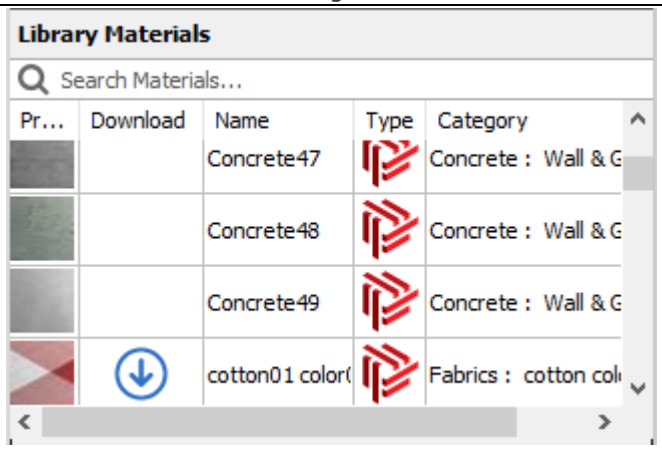
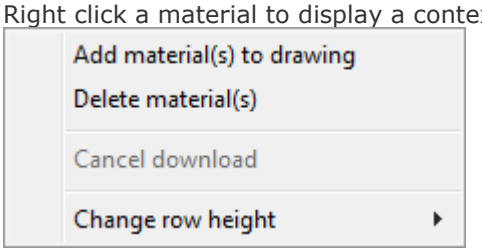
: ***matbrowseropen***

Displays a dialog box:

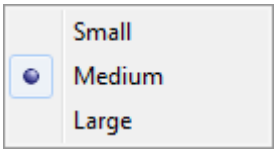


1. Material definitions in the current drawing
2. Search field
3. Library Material (downloaded)
4. Selected Library Material
5. Library Material (not available yet - download needed)
6. Preview of the currently selected library material

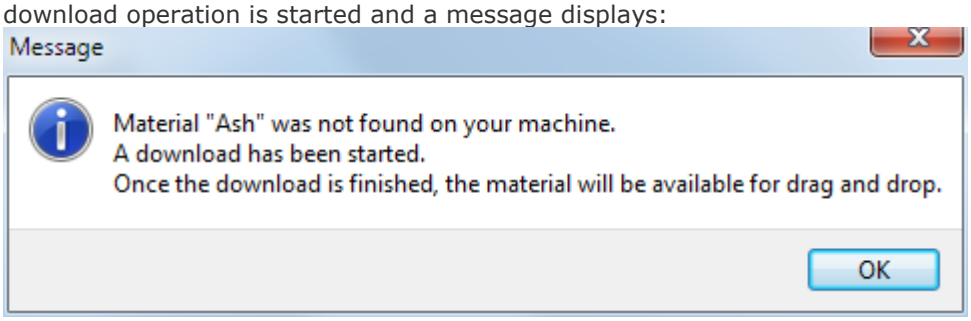
Command Options

Option	Description
Drawing Materials	A list of material definitions contained in the current drawing. Drag and drop a material onto an entity to assign the material. Right click a material to display a context menu:  • Addmaterial(s) to library: copies the selected material(s) to the materials library (= the folder which is defined by the *RenderMaterialPath* user preference). • Change row height: Choose *Small*, *Medium* or *Large* in the context menu. 
Library Materials search field	Allows fast filtering of the material list  Type a search string in the search field. Only materials that contain the search string in their name are listed. The material list is updated as you type. Click the X-button at the right hand side to clear the search field.
Library Materials list	 Click the *Download*, *Name* or *Category* column heading to sort the list accordingly. Click again to reverse the sort order. Right click a material to display a context menu:  • Add material(s) to drawing: copies the selected material(s) to the current drawing. • Delete material(s): removes the selected material(s) from the materials library (= the folder which is defined by the *RenderMaterialPath* user preference). • Cancel download: stops an ongoing material definition download.

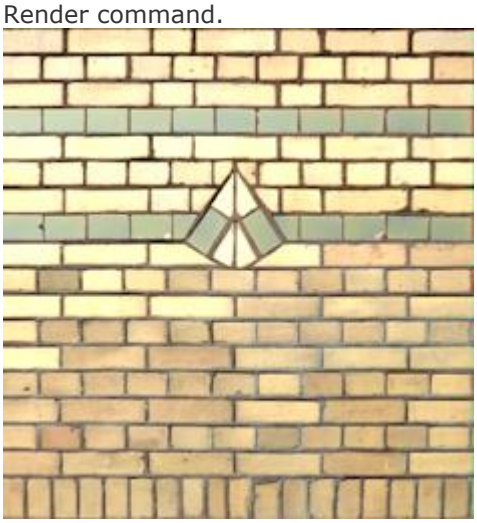
- Change row height: Choose *Small*, *Medium* or *Large* in the context menu.



Download field:
The down arrow indicates that the material is not downloaded yet. Double click to download the material.
Downloaded materials are saved in the folder which is defined by the *RenderMaterialPath* user preference.
(e.g. C:\ProgramData\Bricsys\RenderMaterials).
Drag and drop a material onto an entity to assign the material. The definition is added to the current drawing.
If a not-yet-downloaded Redway material is dragged onto a material, the download operation is started and a message displays:



The Redway high-resolution material definitions are used exclusively by the Render command.



For real-time display, lower-resolution dwg-compatible definitions of the same materials are used.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

MaterialMap - allows to adjust how a texture is mapped to regions, faces and faces of a 3D solid.
MatBrowserClose - closes the materials browser.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

MatchPerspective

Matches the current model space perspective view with a background image by selecting 3 or more matching point pairs.

The command applies to perspective views only.
To toggle the *Perspective* property of the current view do one of the following:

- Click the *Perspective* toolbutton ( ) in the *View* toolbar.
- Choose *Perspective* in the *View* menu.
- Click the *Perspective* toolbutton ( ) in the *View* panel of the *View* Ribbon tab.

Accessing the Command

command bar: *matchperspective*
menu bar: *View | Match Perspective*
toolbar: *View |*

: ***matchperspective***
Prompts you in the command bar:

Enter Model Point: (Specify a point in the 3D model.)

Enter Image Point or [Undo]: (Click the corresponding point in the image.)

Enter Model Point or [Undo]: (Specify a point in the 3D model.)

Enter Image Point or [Undo]: (Click the corresponding point in the image.)

Enter Model Point or [Undo]: (Specify a point in the 3D model.)

Enter Image Point or [Undo]: (Click the corresponding point in the image.)

Enter Model Point or [Undo] <Match>: (Specify more point pairs or press Enter to match the view.)

The model and image point pairs are marked, and connected by green lines.

The perspective matches the background image.

Optionally execute the View command and save the view.

Command Options

Option	Description
Undo	Removes the previous point pair.
Match	Matches the model space view with the background image.

Related Commands

Background - sets the model space background for the current viewport and saved views.
Camera - Creates perspective views.
-View - Creates and sets named views through the command bar.
View - Creates and sets named views through the Drawing Explorer.

MatchProp

Assigns the properties of one entity to one or more other entities (short for "match properties").

Accessing the Command

command bar: *matchprop*
transparent: *'matchprop*
alias: *ma*
toolbar: *Standard* |

: ***matchprop***
Prompts you in the command bar:

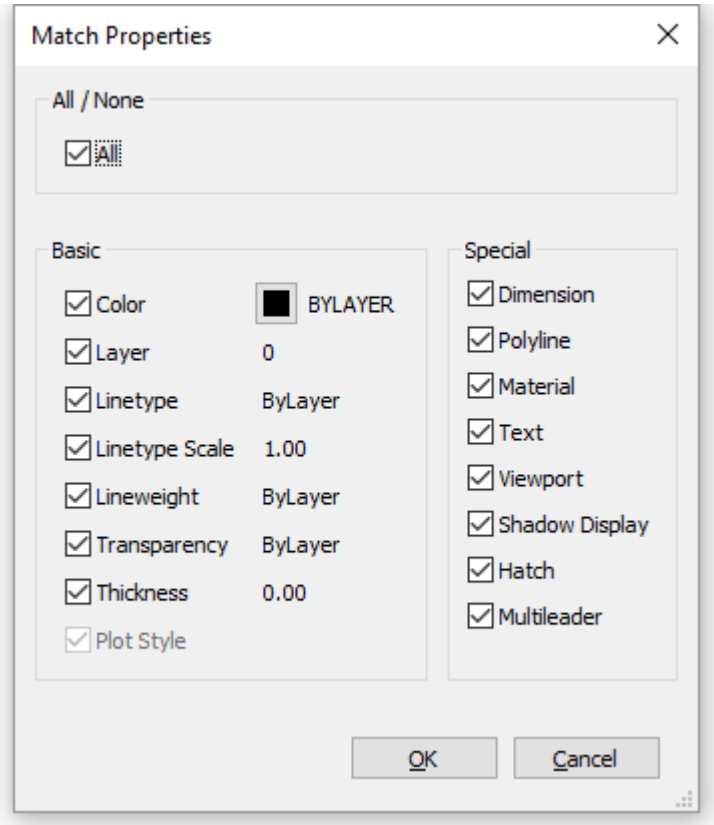
Select entity to copy properties from: (Choose one entity.)
Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader'

Select entities to copy properties to or [Settings]: (Choose one or more entities.)

Select entities to copy properties to or [Settings]: (Choose additional entities, or press Enter to exit the command.)

Command Options

Option	Description
Select entity to copy properties from	Specifies the entity whose properties will be copied. The currently active settings display in the command bar.
Select entities to copy properties to	Specifies the entities whose properties will be changed.
Settings	Sets the active settings. Displays a dialog box:



Click the OK button when done.

You are prompted:

Select entities to copy properties to or [Settings]: Select entities or press S + Enter to reopen the *Match Properties* dialog box.

Procedures

To copy properties between entities

Related Commands

Copy - copies entities.
Properties - changes properties assigned to entities.
Layer - assigns properties to layers.
Color - assigns the working color.
Linetype - assigns the working linetype.
LWeight - assigns the working lineweight.

MaterialMap

Allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.

Accessing the Command

command bar: *materialmap*
menu bar: *View | Rendering | Material Map*
toolbar: *Rendering |*

: ***materialmap***
Prompts you in the command bar:

Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>:
(Choose an option or press Enter to choose Box mapping.)

Select faces or entities: (Select an entity.)

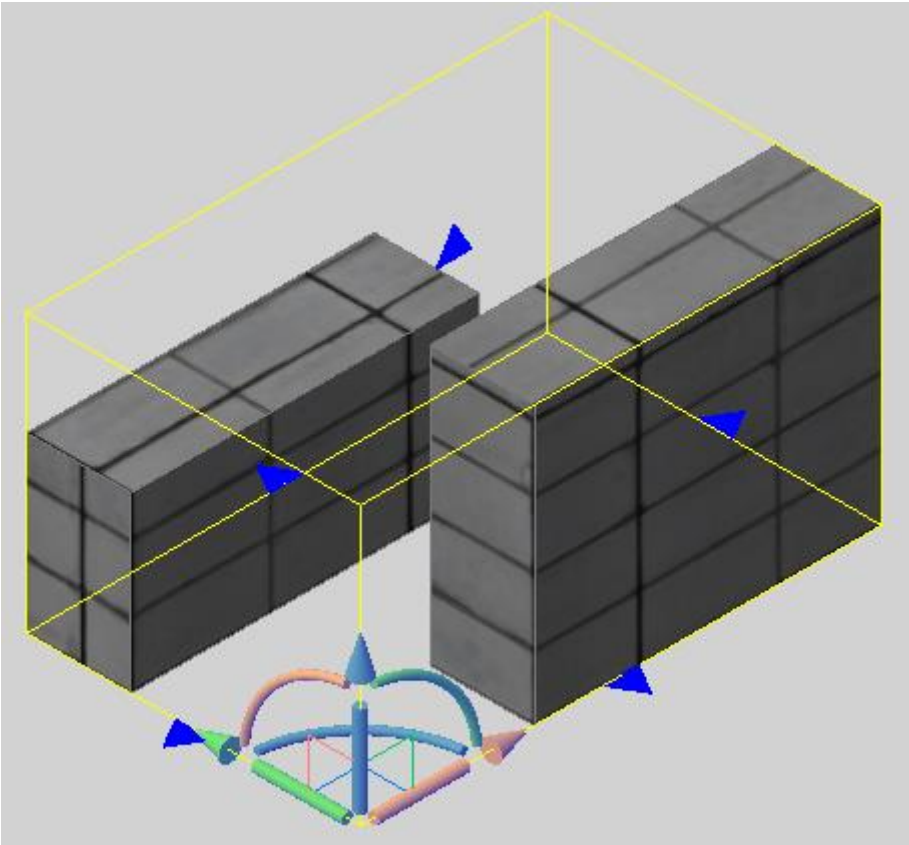
Entities in set: 1

Select faces or entities: (Select more entities or press Enter.)

The manipulator of the selected mapping mode displays.

Command Options

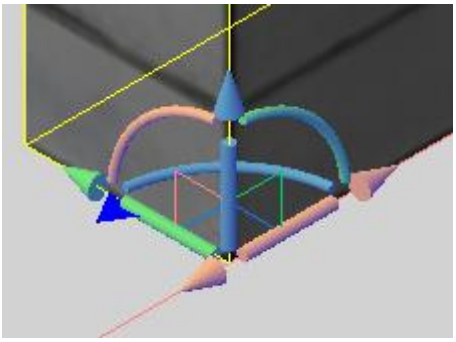
Option	Description
Select faces or entities	Use any of the selection procedures to select entities. See the Select command. All selected entities are included in the yellow bounding box of the manipulator.



To scale the mapping image:
Drag one of the blue arrow grips to scale the texture in the X-, Y- or Z direction.

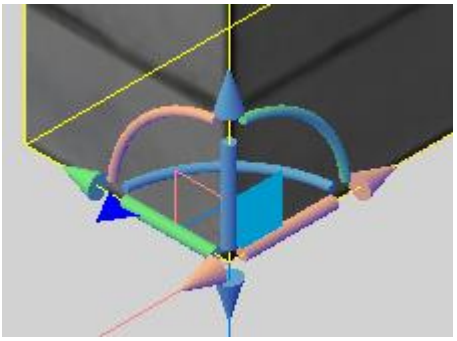
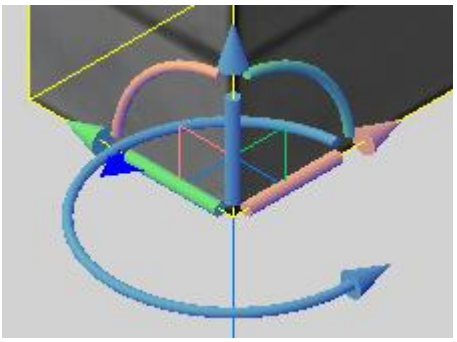
To move the mapping image:
Do one of the following:

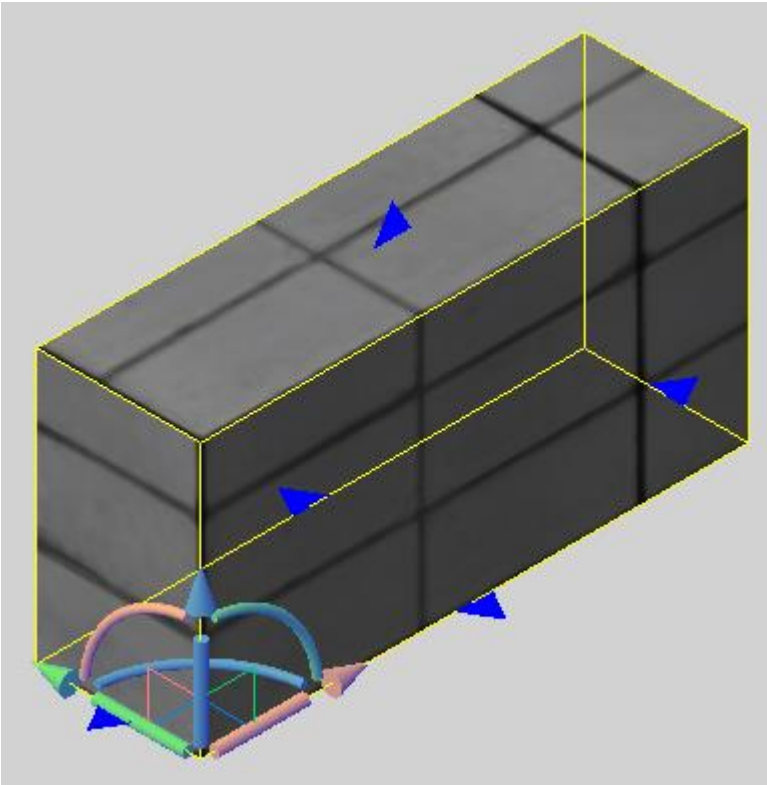
- Place the cursor over the one of the tripod axes and click when a second arrowhead displays on the selected axis, then move the cursor to adjust the position of the pattern in the selected direction.

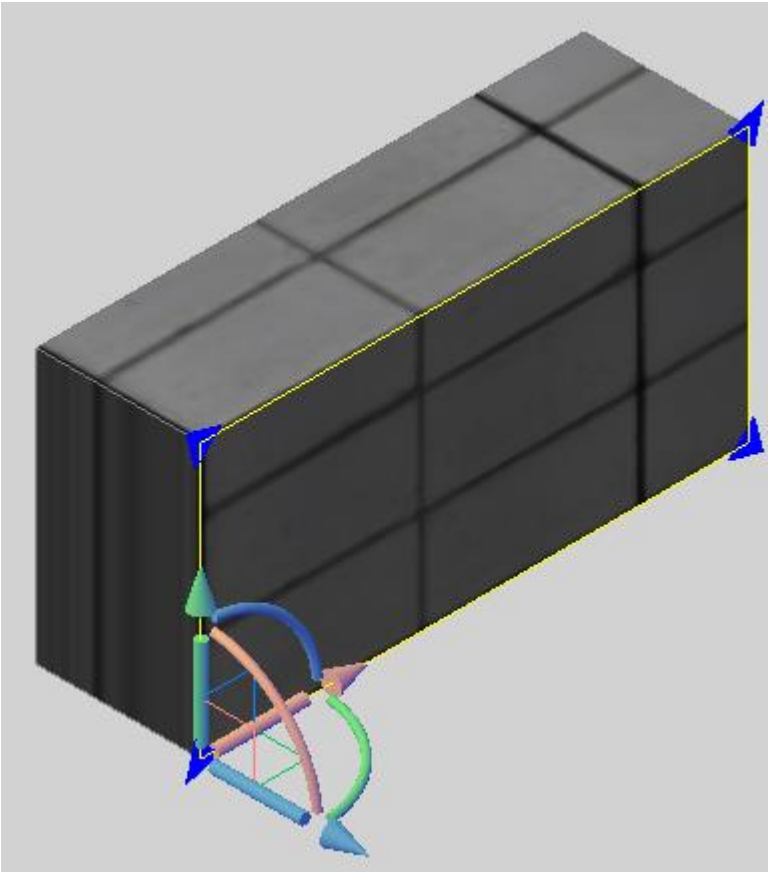
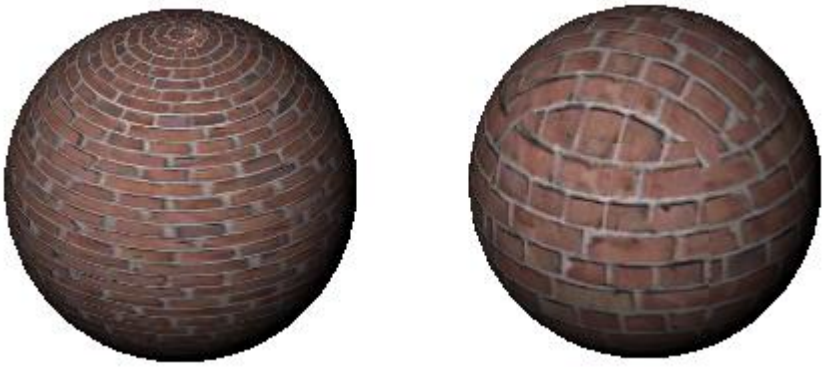
  

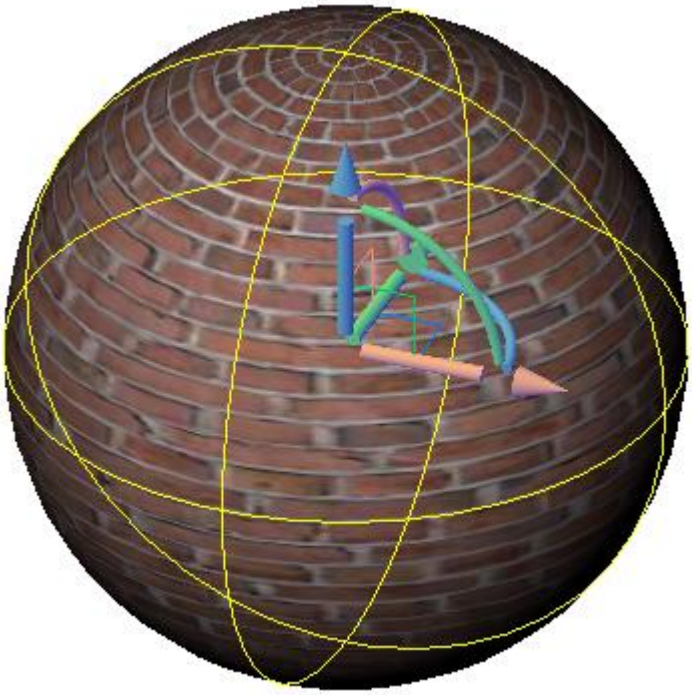
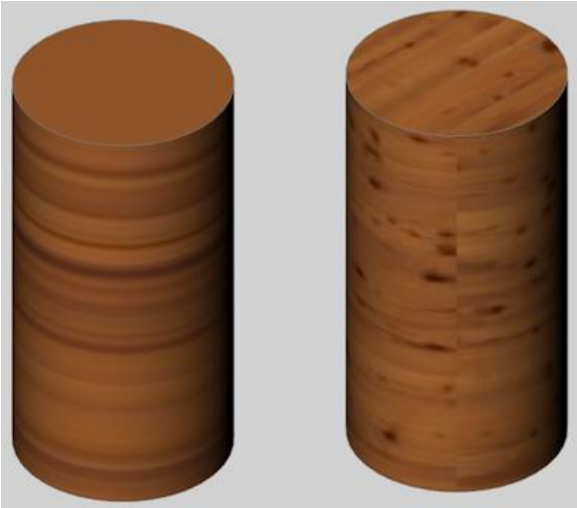
  The box tripod has three axes X (green), Y (red) and Z (blue); in the image above the Y-axis is selected.

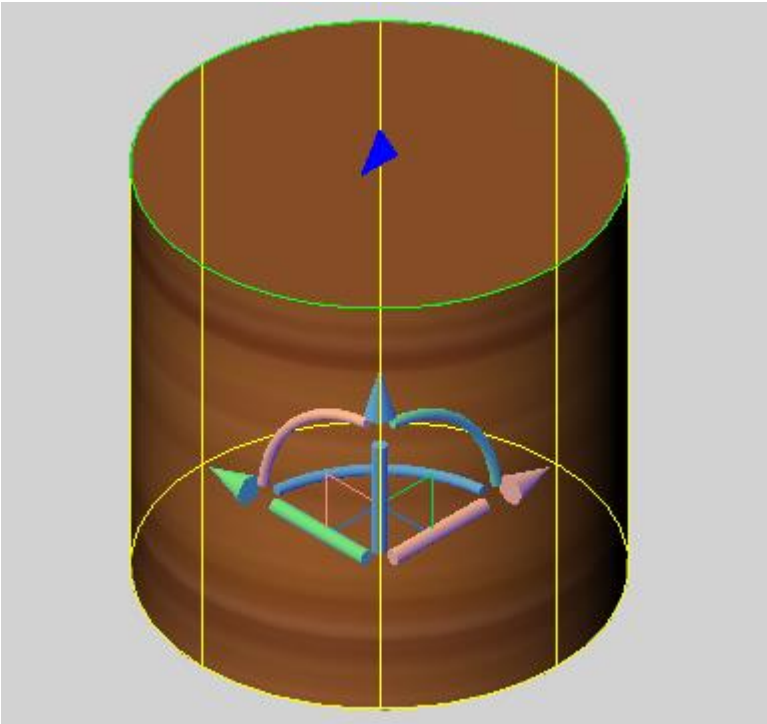
- Place the cursor over a plane quarter (XZ, YZ or XY) and click when the quarter highlights and a second arrowhead displays on the defining axes

	of the plane, then move the cursor to adjust the position of the pattern in the selected plane .  The box tripod has three plane handles: XY, XZ and YZ (left); in the image above the YZ-plane handle is selected. To rotate the mapping image: **Do the following:** • Move the cursor over one of the rotation handles and click when the handle expands. The rotation axis in the color of the selected handle displays. The material map rotates dynamically around the selected axis.  • Click a point to define the rotation angle.
Box	Wraps the texture around the six faces of a the bounding box of the selected entity or entities.

	 Prompts you: Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>: Press Enter to accept the mapping or choose an option. • Reset: Restores the default mapping; reverses all previously applied adjustments. • Switch mapping mode: allows to apply a different mapping mode. Prompts you: Select an option [**Box**/**Planar**/**Spherical**/**Cylindrical**/**Copy mapping to** (Y)/**Reset mapping**)] <Box>: Select an option.
Planar	Aligns the texture to a single face. To select a face of a 3D solid: - Set SELECTIONPREVIEW = 2 or 3 or check the *Select Faces* option of the SELECTIONMODES system variable. - Place the cursor over the 3D solid face and click when the face highlights. Repeatedly hit the TAB key to select an obscured face

	 Prompts you: Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>: Press Enter to accept the mapping or choose an option. • **Reset**: Restores the default mapping; reverses all previously applied adjustments. • **Switch mapping mode**: allows to apply a different mapping mode. Prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>:** Select an option.
Spherical	Aligns the texture to a spherical shape. The top and bottom edges of the image are compressed to a point at the north and south poles of the sphere.  Spherical mapping of a brick pattern (left) compared to box mapping (right).

	 Prompts you: Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>: Press Enter to accept the mapping or choose an option. • **Reset**: Restores the default mapping; reverses all previously applied adjustments. • **Switch mapping mode**: allows to apply a different mapping mode. Prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>:** Select an option.
Cylindrical	Aligns the texture to a cylinder shape. The vertical edges of the image are wrapped together. The height of the image is scaled along the axis of the cylindrical shape.  Cylindrical mapping of a wood pattern (left) compared to a box mapping (right).

	 Prompts you: Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>: Press Enter to accept the mapping or choose an option. • **Reset**: Restores the default mapping; reverses all previously applied adjustments. • **Switch mapping mode**: allows to apply a different mapping mode. Prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>:** Select an option.
Copy mapping to	Applies mapping from the source entity or face to the selected entities, thus duplicating the mapping, including any adjustments, to other entities. All entities share the same mapping origin, axes and scale. As a result the texture image is extended seamlessly from one entity to another . Prompts you: Select face or entity: Select the source entity. Select the entities to copy the mapping to: Select faces or entities: Select the target entity. Entities in set: 1 Select faces or entities: Select more target entities or press Enter.
Reset mapping	Restores the default mapping; reverses all previously applied adjustments.

Related Commands

MatBrowserOpen - opens the materials browser, which shows an overview of available high-resolution render materials.
Materials - creates materials and edits their properties through the Drawing Explorer.

Materials

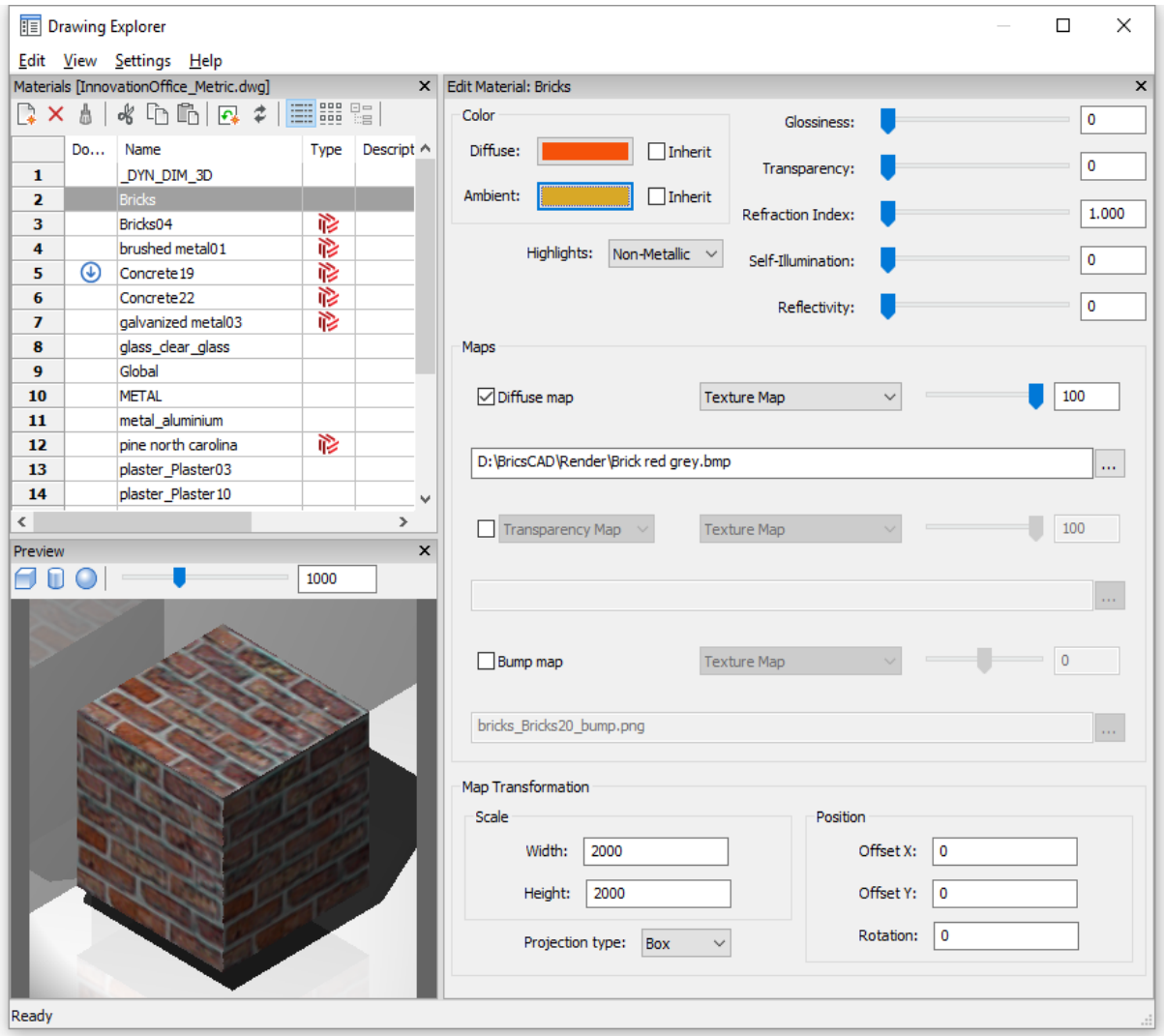
Creates materials and edits their properties through the Drawing Explorer.

Accessing the Command

command bar: *materials*
menu bar: *Tools | Drawing Explorer | Materials ...*
toolbar: *Rendering*|

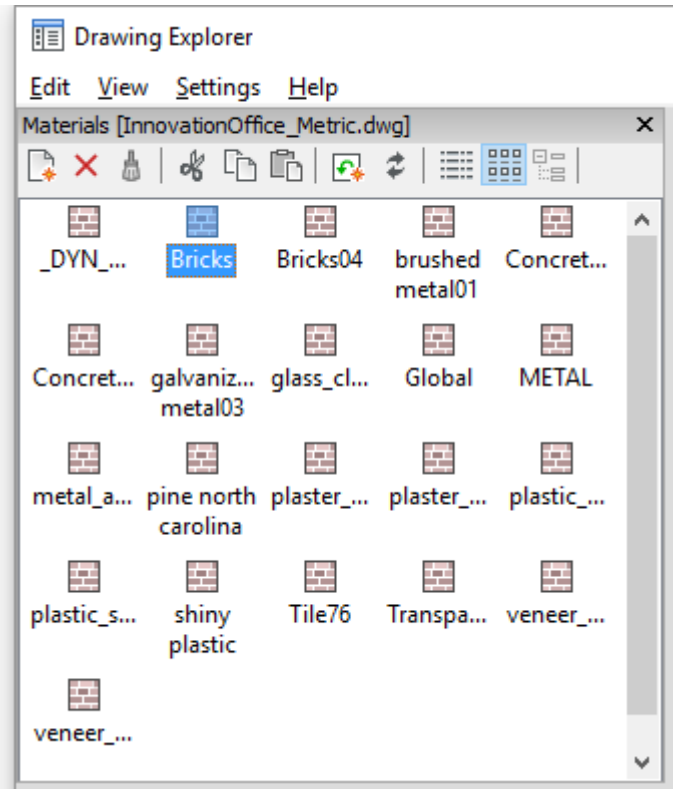
: ***materials***

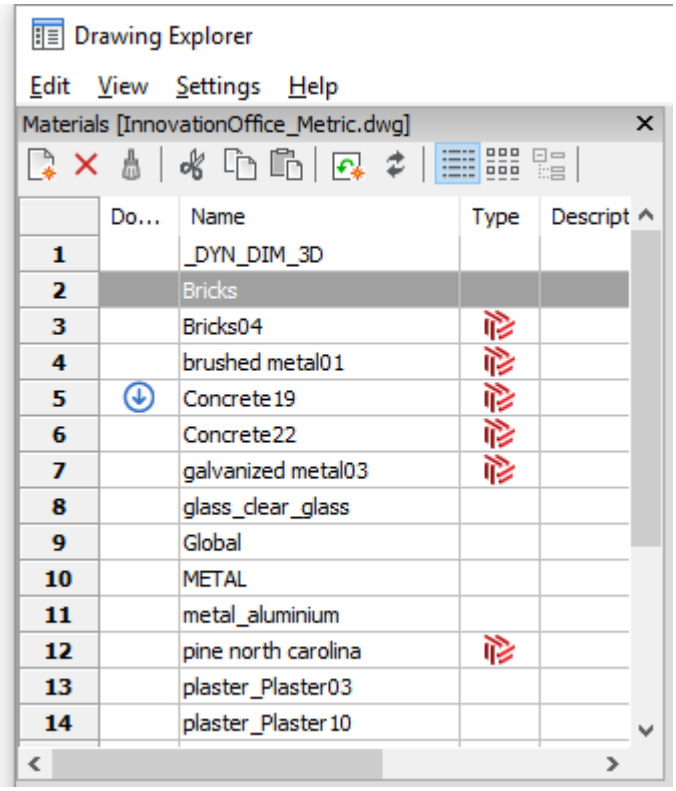
Displays the Materials section of the Drawing Explorer dialog box:



RedWay materials (), which are imported in the Materials Browser, cannot be edited. All settings for such materials are greyed out, except for the *Map Transformation* settings.
Right click a Redway material, then choose *Convert to regular material* in the context menu to activate all settings fields.

Command Options

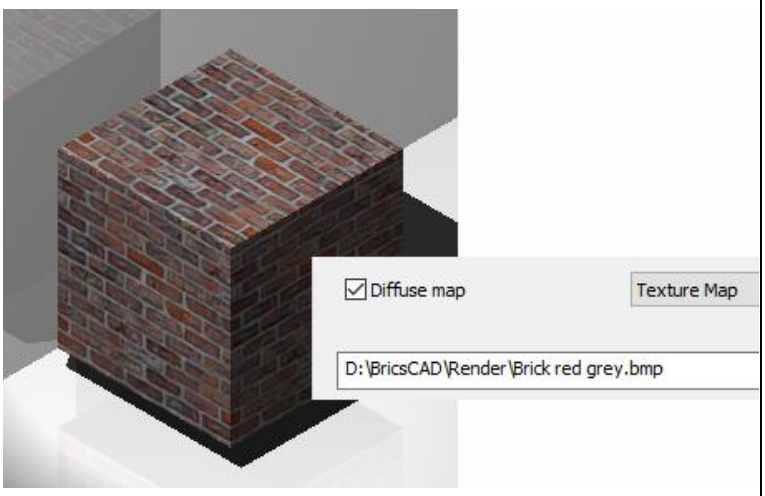
Option	Description
Details	
New	Creates a new material. Click the icon, then name the material and optionally fill out the description field (*Detail View* only).
Delete	Deletes the selected material(s). The *Global* material cannot be deleted.
Icon View	Displays the *Icon View* of the materials:  Select a material to edit its properties. Click the material name to rename.

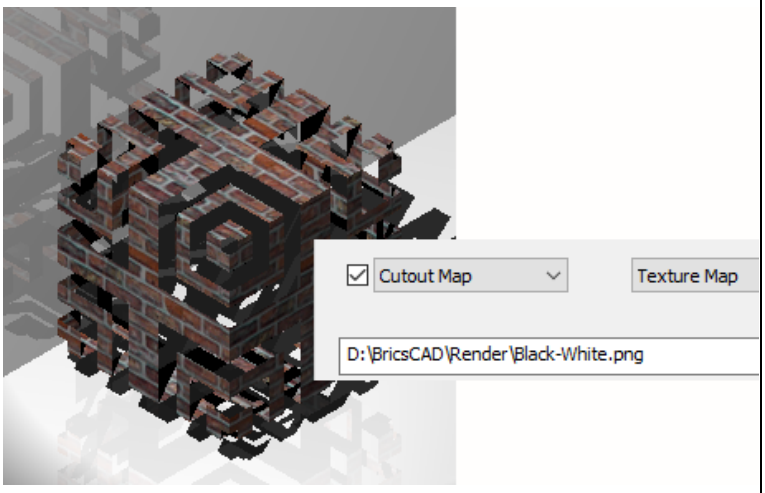
Detail View	Displays the *Detail View* of the materials:  Select a material to edit its properties. Click the *Material Name* field to rename. Click the *Description* field to add or edit the material description. Double click the *Download* icon ( ) in the *Download* colums to download Redway materials ( ).
Preview	
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. Do one of the following: • Drag the slider. • Click before or after the slider.
Edit Material	
Name	Type a name for the material. The material name must be unique.

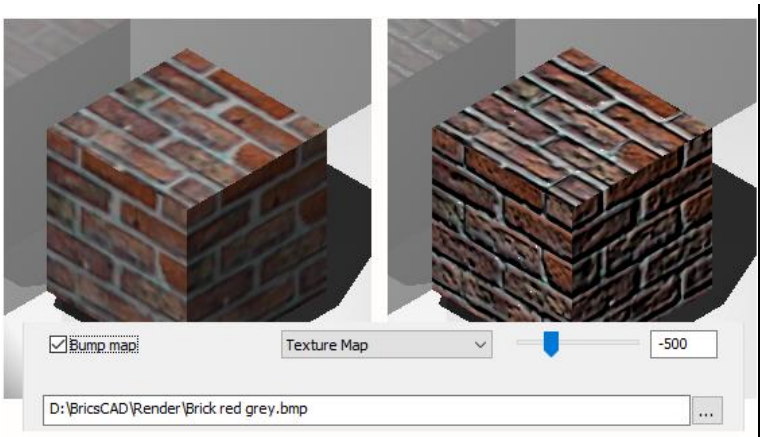
	The *Global* material cannot be renamed.
Description	Type a description for the material. The material description is optional.
Color	
Diffuse Ambient	Sets the diffuse and ambient color. Do one of the following: • Check the *Inherit* option to apply the entity color. • Click the colored tile, then choose the color in the Color dialog box.
Highlights	Sets the highlight property of the material. Click the button then choose either *Non-Metallic* or *Metallic*.

Glossiness
Transparency
Refraction Index
Self-Illumination
Reflectivity
	Defines a property of the material surface.
	Do one of the following:
	• Drag the slider.
	• Click before or after the slider.
	• Type the desired value in the settings field.
	  • Glossiness: range = 0 - 100
	  • Transparency: range = 0 - 100
	  • Refraction Index: range = 1.00 - 3.00
	  • Self-Illumination: range = 0 - 100
	  • Reflectivity: range = 0 - 100

Maps	Texture maps add detail to a surface which is not included in the 3D model itself.
	To apply a texture map:
	• Check the *Diffuse, Transparency* or *Bump Map* option.
	• Do one of the following:
	  • Select *Wood* or *Marble* from the *Texture Map* option button.
	  • Choose *Texture Map*, then press the *Browse* button (...) and select a texture bitmap file (*).
	• Set the *Blend factor* for the texture map.
	Diffuse and Transparency maps: range = 0 / 100.
	Bump map: range = -1000 / +1000
	Use the slider or type the factor in the settings field.
	The *Blend factor* sets the intensity of the texture map.
	(*) The TextureMapPath user preference defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under *Textures*, each containing a number of texture files of the same name. Images in folder *1* have a size of 256 x 256 pixels, folder *2* contains images of 512 x 512 pixels, images in folder *3* have a size of 1024 x 1024 pixels. If the *Diffuse map*

	setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the *TextureMapPath* user preference to folder *1*, *2* or *3*.
Diffuse Map	Diffuse maps apply a texture to the surface of a material such as wood grains, bricks or tiles. Drag the slider to blend the texture map with the Color settings. 
Transparency Map Transparency Map Cutout Map Cutout Map Inverted	The selected *Texture Map* can be applied as *Transparency Map* or as *Cutout Map*. In the examples below, this image is used as transparency map and cutout map: If *Transparency Map* is selected, the selected *Texture Map* image defines a transparency map. The image is recommended to be a grayscale image. White pixels will be invisible, black pixels will be opaque and gray pixels will be transparent. Notes: • Colored images are converted to grayscale images in the background. • The meaning of black and white is the inverse of *Cutout Map* textures. • The alpha channel of the image is ignored. **Blend factor**: the transparency values in the texture will approach the scalar *Transparency* setting as the blend factor approaches 0. This means that if the *Blend factor* is 100, then the transparency value will controlled completely by the *Texture Map* image. If the *Blend factor* is 0, the transparency texture will be completely ignored and the scalar *Transparency* value will be used.

	 If *Cutout Map* is selected, then the selected transparency map defines cutouts: parts of the objects will be fully visible and parts not visible at all, depending on the pixels of the selected image. The image is recommended to be a bitonal black-and-white image, without grays. White pixels are visible, and black pixels are invisible. The *Cutout Map Inverted* option inverts the effect: visible parts become invisible and vice versa. Notes: • Gray scale and color images are converted to black-and-white images in the background. • The meaning of black and white is the inverse of *Transparency Map* textures. • The alpha channel of the image is ignored. 
Bump Map	Bump mapping is a technique for simulating bumps and wrinkles on the surface of an object. The result is an apparently bumpy surface although the surface of the underlying object is not modified. Blend factor range = -1000 / +1000

	 Left: Diffuse Map only; Right: Bump Map applied.
Map Transformation	
Scale	The texture map image is applied at a size of 1 drawing unit multiplied by the *Width* and *Height* factors. E.g. if the *Width* and *Height* factors or both set to 10, the size of the texture image is 10 x 10 drawing units.
Position	
Offset	Texture maps maps are tiled starting from the origin of the WCS. To adjust the tiling you can define an Offset X and Offset Y. The offsets are expressed in drawing units.
Rotation	Sets the rotation of the texture maps. Type the desired rotation in the *Rotation* field.

Related Commands

MatBrowserOpen - opens the materials browser, which shows an overview of available high-resolution render materials.
MaterialMap - allows to adjust how a texture is mapped to regions, faces and faces of a 3D solid.
Render - calculates a photorealistic image of the 3D model.
Light - creates a new light in the drawing.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

Explorer - opens the Drawing Explorer dialog box.

MatLib

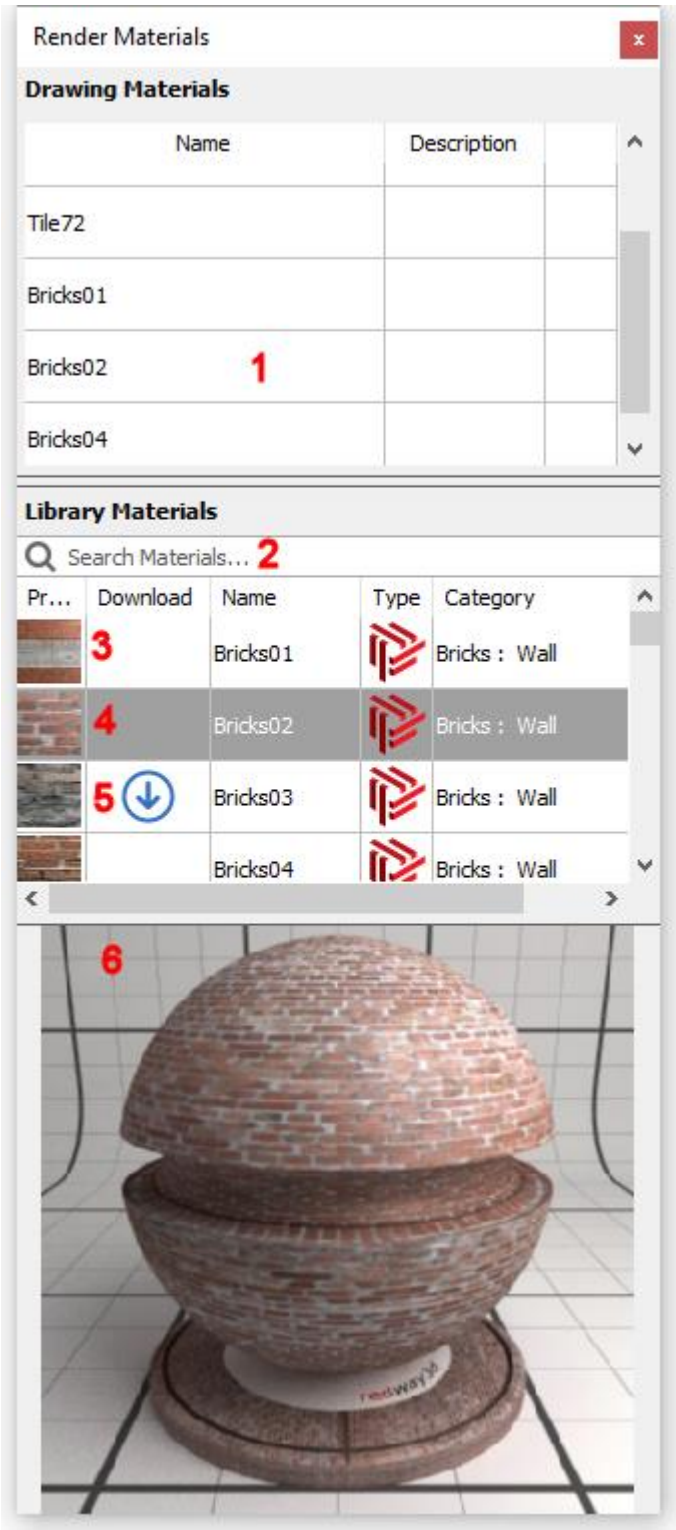
Opens the materials browser, which shows an overview of available high-resolution render materials.

Accessing the Command

command bar: *matbrowseropen*
aliases: *matb, matlib*

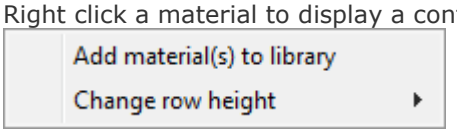
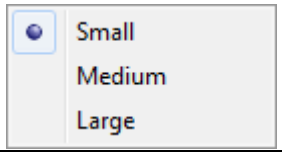
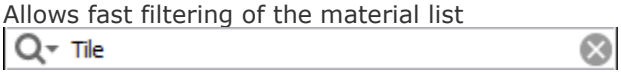
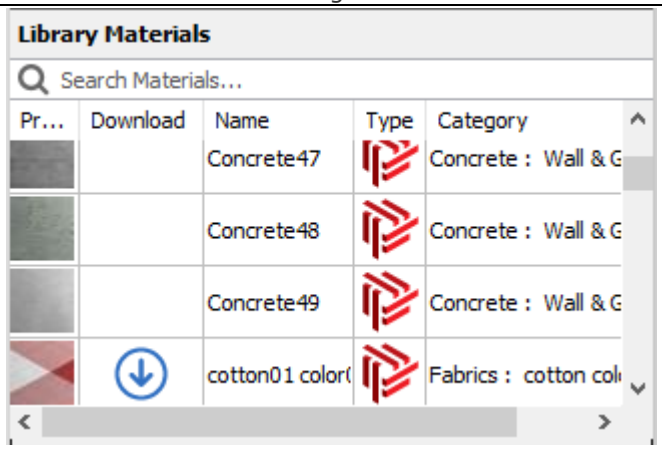
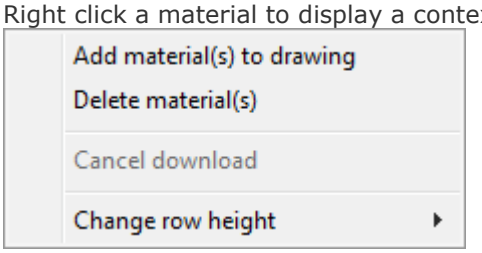
: ***matbrowseropen***

Displays a dialog box:

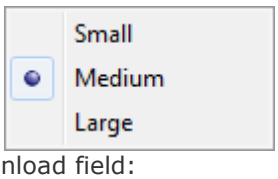


1. Material definitions in the current drawing
2. Search field
3. Library Material (downloaded)
4. Selected Library Material
5. Library Material (not available yet - download needed)
6. Preview of the currently selected library material

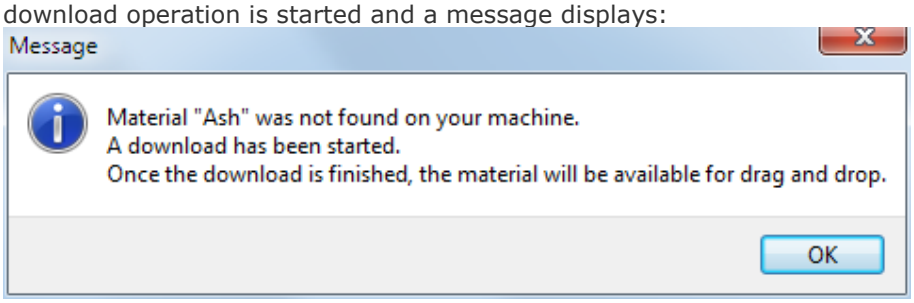
Command Options

Option	Description
Drawing Materials	A list of material definitions contained in the current drawing. Drag and drop a material onto an entity to assign the material. Right click a material to display a context menu:  • Addmaterial(s) to library: copies the selected material(s) to the materials library (= the folder which is defined by the *RenderMaterialPath* user preference). • Change row height: Choose *Small*, *Medium* or *Large* in the context menu. 
Library Materials search field	Allows fast filtering of the material list  Type a search string in the search field. Only materials that contain the search string in their name are listed. The material list is updated as you type. Click the X-button at the right hand side to clear the search field.
Library Materials list	 Click the *Download*, *Name* or *Category* column heading to sort the list accordingly. Click again to reverse the sort order. Right click a material to display a context menu:  • Add material(s) to drawing: copies the selected material(s) to the current drawing. • Delete material(s): removes the selected material(s) from the materials library (= the folder which is defined by the *RenderMaterialPath* user preference).

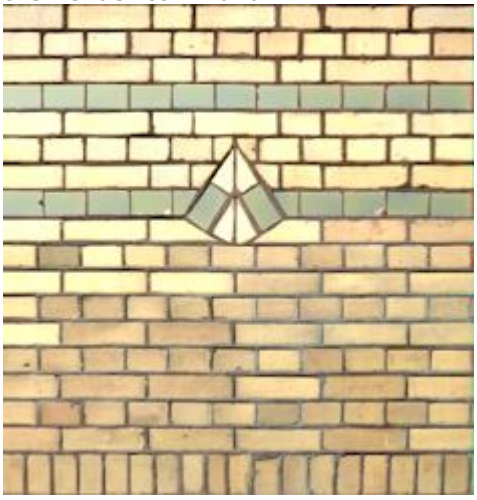
- Cancel download: stops an ongoing material definition download.
- Change row height: Choose *Small*, *Medium* or *Large* in the context menu.



Download field:
The down arrow indicates that the material is not downloaded yet. Double click to download the material.
Downloaded materials are saved in the folder which is defined by the *RenderMaterialPath* user preference.
(e.g. C:\ProgramData\Bricsys\RenderMaterials).
Drag and drop a material onto an entity to assign the material. The definition is added to the current drawing.
If a not-yet-downloaded Redway material is dragged onto a material, the download operation is started and a message displays:



The Redway high-resolution material definitions are used exclusively by the Render command.



For real-time display, lower-resolution dwg-compatible definitions of the same materials are used.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

MaterialMap - allows to adjust how a texture is mapped to regions, faces and faces of a 3D solid.
MatBrowserClose - closes the materials browser.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Measure

Places points or blocks along entities.

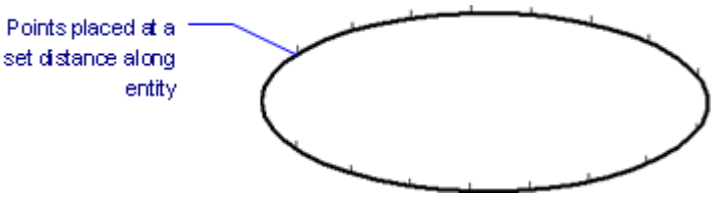
Accessing the Command

command bar: *measure*
menu bar: *Modify | Measure*
toolbar: *Modify | Measure/Divide |*

: ***measure***
Prompts you in the command bar:

Select entity to measure: (Choose a line, polyline, spline, arc, circle, or ellipse.)

Segment length [insert Block]: (Enter a number, or type B.)



Command Options

Option	Description
Select entity to measure	Chooses a single entity: a line, polyline, spline, arc, circle, or ellipse.
Segment length	Specifies the distance between segments; the segments begin at the pick point. Use the PdMode system variable to make the points visible.
Insert block	Uses blocks in place of points; prompts you: Name of block to insert [? to list]: Specify the name of the block; the block must exist in the drawing. Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-Align blocks>: Choose an options.
Name of Block to Insert	
Align Blocks with Entity?	Toggles whether blocks are aligned with the entity: • Yes - rotates blocks to match the entity. • No - does not rotate blocks; they maintain their orientation.
Segment length	Specifies the distance between segments; the segments begin at the pick point.

Procedures

To measure an entity using points
To place blocks a specified interval along an entity

Related Commands

Divide - places evenly-spaced points along entities.
Point - places points in drawings.
DdPType - specifies the look and size of point entities through the Settings dialog box

Menu

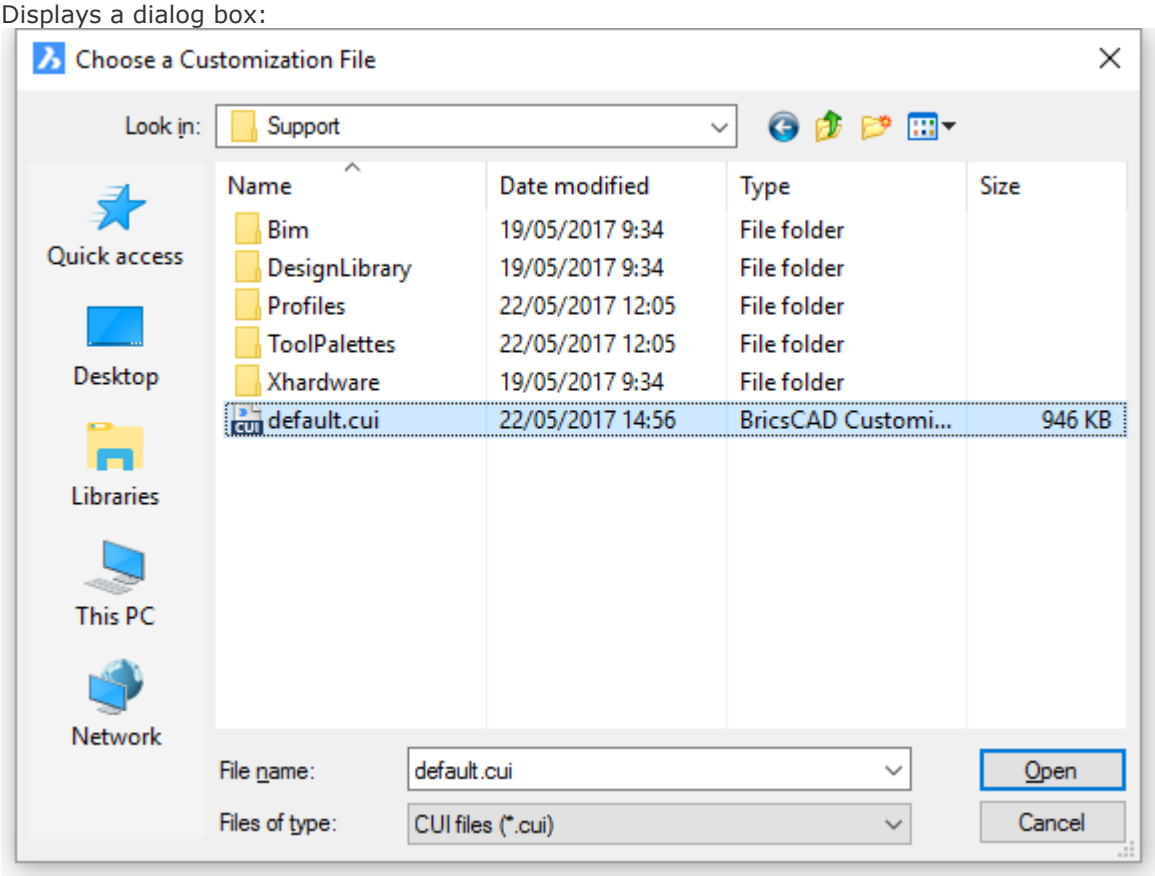
Loads menu files into BricsCAD's user interface.

Accessing the Command

command bar: *menu*
menu bar: *Tools | Load menu*
toolbar: *Tools |*

: ***menu***

Displays a dialog box:



Choose a menu file, and then click Open.

Command Options

Option	Description
File name	Specifies the name of the menu file.
File type	Specifies the format of menu file: • CUI - compatible with AutoCAD 2007 and later. • ICM - compatible with IntelliCAD menu files. • MNU - compatible with AutoCAD 2006 and earlier. • MNS - LISP source code used with MNU files.
Open	Opens the menu file.

	Caution: loading a menu file can replace the user interface of BricsCAD.

Related Commands

CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
Menu - loads menu files into BricsCAD's user interface.
MenuLoad - loads CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files.
MenuUnload - unloads menu files.

MenuLoad

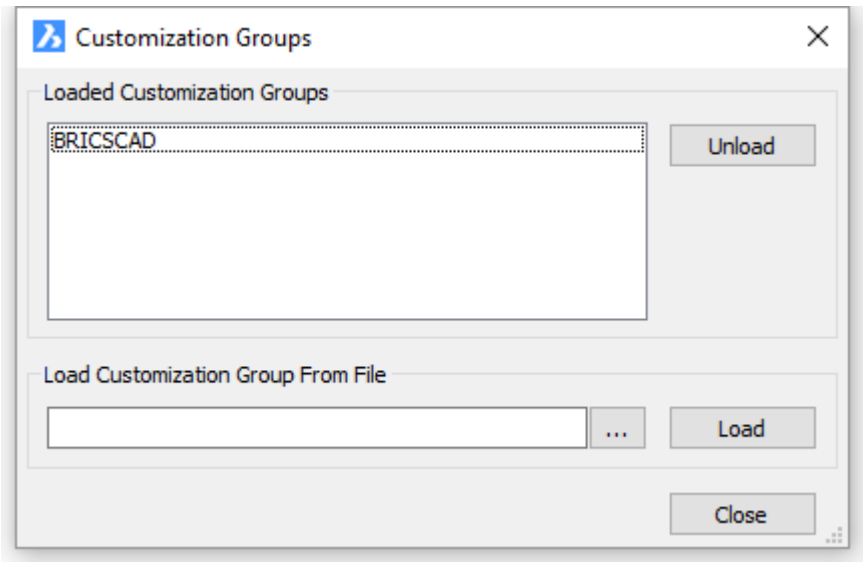
Loads CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files.

Accessing the Command

command bar: *menuload*
alias: *cuiload*

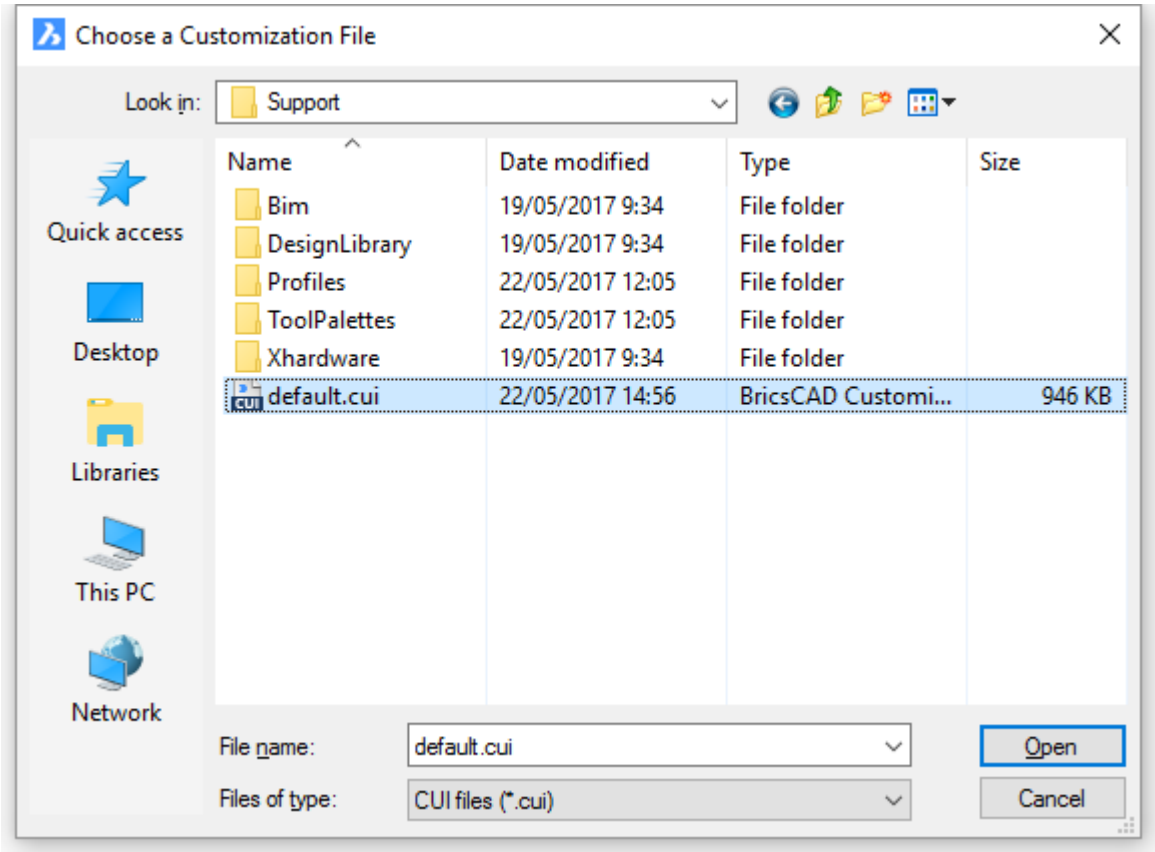
: ***menuload***

Displays a dialog box:



Click the **...** button.

The next dialog box appears:



Choose a CUI, MNU, MNS, or ICM file, and then click Open.
*The Customization Groups dialog box reappears. Click **Load** to add the customization file to the user interface.*

Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.
...	Displays the *Choose a Customization File* dialog box.
Load	Loads the opened customization file.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Commands - lists the names of internal and external commands.
CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
Menu - loads menu files into BricsCAD's user interface.
MenuUnload - unloads menu files.
Netload - loads a .NET application.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

MenuUnload

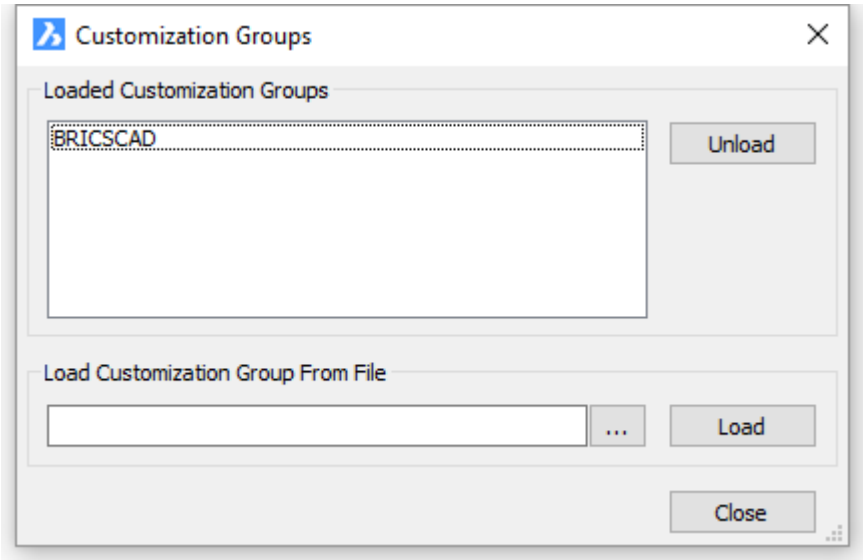
Unloads customization groups from the user interface.

Accessing the Command

command bar: *menuunload*
alias: *cuiunload*

: ***menuunload***

Displays a dialog box:



Choose a customization group, and then click **Unload**.

Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.
...	Displays the *Choose a Customization File* dialog box.
Load	Loads the opened customization file.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
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VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.

VbaUnload - unloads VBA projects.

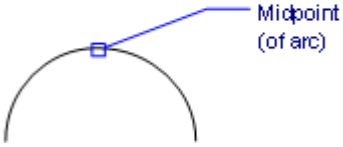
Midpoint

Toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and edges of 3D solids.

Accessing the Command

command bar: *midpoint*
transparent: *mid*
toolbar: *Entity Snaps* |

: ***midpoint***
Displays no prompt in the command bar.



When midpoint entity snap is on, the cursor snaps to the midpoints of entities.

This entity snap works with 3D solid models.

Command Options

Option	Description
Midpoint	Entering the command toggles Midpoint entity snap. It snaps to the midpoint of entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.

Tangent - toggles tangent snap mode.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.

MInsert

Inserts a block as a rectangular array; combines the -Insert and -Array commands (short for "multiple insertion"). Creates a MInsert Block entity.

Accessing the Command

command bar: *minsert*

: ***minsert***
Prompts you in the command bar:

? to list blocks in drawing/~ to open the file dialog/<Block to insert>: (Type the name of a block, press Enter to accept the name of the previously inserted block, or choose an option.)

Insertion point for block or [Multiple blocks/Scale]: (Pick a point to insert the block, or choose an option.)

[Corner] <1.000000>: (Pick a point, type a value to define the X scale factor, or press Enter to accept the default.)

Y scale factor: < Equal to X scale (2.000000)>: (Pick a point, type a value to define the Y scale factor, or press Enter to set the Y scale equal to the X scale.)

Rotation angle for block <0>: (Pick a point, specify a rotation angle, or press Enter to accept the default.)

Number of rows in the array **<1>:** (Enter the number of rows.)

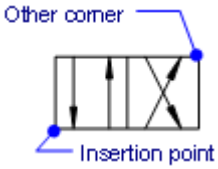
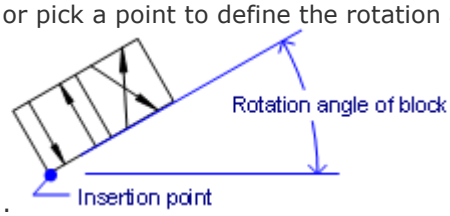
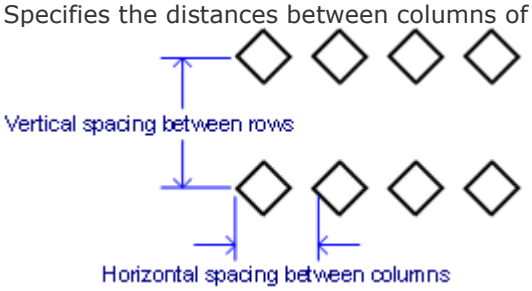
Number of columns **<1>:** (Enter the number of columns.)

Vertical distance between rows**, or** spacing rectangle**:** (Specify the distance between rows, or pick two points.)

Horizontal distance between columns**:** (Specify the distance between columns.)

Command Options

Option	Description
? to list blocks in drawing	Type **?** to list the names of all blocks in the current drawing. Prompts you: **Block(s) to list [* for all]** - press Enter to list all blocks, or enter part of a name to view some blocks. If necessary, press **F2** to view the full list. See TextScr command.
~ to open the file dialog	Type **~** the Insert Block dialog box. Choose a DWG file, and then click **Open**.
Block to insert	Specifies the name of the block to be inserted.
Multiple blocks	This allows you to insert the same block many times quickly. Repeats the following prompt a the end of the command procedure until you press Enter or Esc: **Insertion point for next block** - specify a point.
Scale	Allows to set the X and Y scale factor equally; prompts you: **Scale factor for block** - type a value.
Corner	Allows to specify the scaling of the block by specifying a second corner of a rectangle. The insertion point is the first corner. Prompts you: **Select other corner for scale or enter X scale factor** - pick a second corner, or enter a number for the x scale factor.

	You can pick points or enter x,y coordinates. 
Y scale factor	Specifies the scale factor of the block in the y direction. Enter a negative number to mirror the block. Press Enter to make the y scale = x scale.
Rotation angle for block	Specifies the rotation angle of the block about its insertion point. Enter a value or pick a point to define the rotation angle.  .
Number of rows in the array	Specifies the number of rows; for a linear array, enter 1. To draw the array downwards, enter a negative number.
Number of columns	Specifies the number of columns. To draw the array to the left, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of blocks.
Spacing rectangle	Specifies two points that indicate the spacing between rows and columns.
Horizontal distance between columns	Specifies the distances between columns of blocks. 

Procedures

To insert multiple instances of a block in a rectangular array

Related Commands

3dArray - creates 3D arrays.
Array - creates 2D arrays.
ArrayPath - distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - distributes entity copies into any number of rows, columns, and levels.

Block - creates blocks.
Insert - inserts blocks.

Mirror

Mirrors entities about a mirror line in a 2D plane.

Accessing the Command

command bar: *mirror*
alias: *mi*
menu bar: *Modify | 2D Mirror*
toolbar: *Modify |*

: ***mirror***
Prompts you in the command bar:

Select entities to mirror: (Choose one or more entities.)
Entities in set: 1

Select entities to mirror: (Choose more entities, or press Enter to continue.)

Start of mirror line: (Pick a point, or enter x,y coordinates.)

End of mirror line: (Pick another point.)

Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: (Type Y or press Enter to accept the default option: keep the original entities.)

Command Options

Option	Description
Start of mirror line	Specifies the start point for the line about which entities are mirrored. This line is temporary; as soon as you pick the second point, it disappears. To mirror vertically or horizontally, hold down the Shift key when specifying the second point or use Polar Tracking.
End of mirror line	Specifies the second point of the mirror line.
Delete the original entities?	Determines whether the original entities are erased: • Y - erases the original entities, leaving only the mirrored ones. • N - does not erase the original entities, leaving them in place.

Procedures

Mirroring entities about a line
Mirroring entities about a plane
To mirror entities using grips

Related Commands

Mirror3D - mirrors entities in 3D space.
InsertAligned - inserts mirrored blocks.

Mirror3D

Mirrors entities about a mirror plane in 3D space.

Accessing the Command

command bar: *mirror3d*
alias: *3dmirror*
menu bar: *Modify | 3D Mirror*
toolbar: *Modify |*

: ***mirror3d***
Prompts you in the command bar:

Select entities: (Choose one or more entities.)

Entities in set: 1

Select entities: (Choose more entities, or press Enter to continue.)

Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>: (Enter an option.)

Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: (Type Y or N.)

Command Options

Option	Description
Select entities	Chooses one or more entities to mirror.
Entity	Defines the mirror plane by extrusion direction; prompts you: Pick an entity that has an extrusion direction - select entities with a thickness. This option does not work with flat entities or 3D solids. To give an entity an extrusion, use the Properties command's Thickness option.
Last	Reuses the last mirror plane; does not prompt you.
View	Defines the mirror plane as the current view plane; prompts you: Point on view plane <0,0,0> - specify the x,y,z coordinates of the view plane.
Zaxis	Defines the mirror plane by the z axis and a point on the view plane; prompts you: Point on view plane - defines the view plane by a point. Point on the z-axis (normal) of the plane - specify the height of the view plane.
XY	Specifies the mirror plane as the x,y plane; prompts you: Point on XY plane <0,0,0> - specify the location of the plane through x,y,z coordinates.
YZ	Specifies the mirror plane as the y,z plane; prompts you: Point on YX plane <0,0,0> - specify the location of the plane through x,y,z coordinates.
ZX	Specifies the mirror plane as the z,x plane; prompts you: Point on ZX plane <0,0,0> - specify the location of the plane through x,y,z coordinates.

3points	Defines the mirror plane by three points; prompts you: Second point on plane - specifies the second point; the first point is specified by the 3points prompt. Third point on plane - specifies the third point. Three points define a 2D plane in space.
Delete the original entities?	Determines whether the original entities are erased: • Y - erases the original entities, leaving only the mirrored ones. • N - does not erase the original entities, leaving all in place.

Related Commands

Mirror - mirrors entities about a mirror line in a 2D plane.
InsertAligned - inserts mirrored blocks.

MLeader

Creates multileader entities using the current multileader style.

Accessing the Command

command bar: *mleader*
menu bar: *Dimension | Multileader*
toolbar: *Dimensions |*

: ***mleader***

Prompts you in the command bar:

Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:
Specify the location of the arrowhead or press Enter to choose an option.

A multileader entity consists of three components: a leader line with an arrowhead, a landing and the leader content (either a multiline text entity or a block).

Command Options

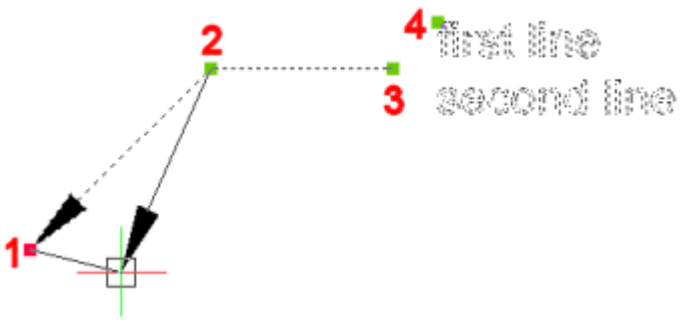
Option	Description
Arrowhead first (default)	Prompts you: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>: (Pick a point or choose an option.) *Specify leader landing location: (Pick a point.)* If the command is terminated here, no text is associated with the multir leader entity. The Mtext command is launched.
Leader Landing first	Prompts you: Specify leader landing location or [leader arrowHead first/Content first/Options] <Options>: (Pick a point or choose an option.) Specify leader arrowhead location: (Pick a point.) If the command is terminated here, no text is associated with the multir leader entity. The Mtext command is launched.
Content First	Prompts you: Specify first corner of text or [leader arrowHead first/leader Landing first/Options] <Options>: (Pick a point or choose an option.) Specify the opposite corner: (Pick a point.) The Mtext command is launched. Type the text in the multiline text editor, then press the *OK* button or click outside the multiline text editor. Specify leader arrowhead location: (Pick a point.)
Options	Sets overrides to the current multileader style. Prompts you: Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>: • Leader type: Sets the leader type: straight, spline or none. Prompts you: Select a leader type [Straight/sPline/None] <Straight>: Choose an option.

	• Leader landing: Specifies whether to add a leader landing or not. Prompts you: Use landing [Yes/No] <Yes>: Choose an option. • Content type: Specifies the content type of the multileader: multiline text, block or none. Prompts you: Select a content type [Block/Mtext/None] <Mtext>: Choose an option. • Maxpoints: Specifies the maximum leader line nodes. Prompts you: Enter the maximum points for leader line <2>: Enter a value in the command bar. • First angle: Constrains the first leader segment. Prompts you: Enter first angle constraint <0>: Type a value in the command bar. The angle of the first leader segment is constrained to this value or a multiple of this. • Second angle: Constrains the second leader segment. Prompts you: Enter second angle constraint <0>: Type a value in the command bar. The angle of the second leader segment is constrained to this value or a multiple of this. • Exit options: Terminates the definition of overrides and prompts you to specify the location of the multi leader entity.

Grips Editing

MLeaders can be edited through grips.

1. Select the MLeader in the drawing.
   For a single segment leader entity, four grips display: Endpoint (1), Vertex (2), Landing (3) and Text (4).
2. Drag the grips to perform the following actions:
   - Move the arrowhead (1)
   - Move the landing and the mleader content (2)
   - Edit the landing length(3)
   - Move text (4)



Procedures

To create a multileader
Adding vertices to a leader line of a multileader
Removing vertices from a leader line of a multileader
Adding leader lines to a multileader
Removing leader lines from a multileader

Related Commands

MLeaderEdit - adds leader lines to and removes leader lines from a multileader entity.
MLeaderStyle - Creates and manages multileader styles in the Drawing Explorer dialog box.

BimTag - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.
BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
Dimleader - places leaders with text, blocks, entities, or tolerances.
Leader - places leaders with text, blocks, entities, or tolerances.
QLeader - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

MLeaderEdit

Adds leader lines to and removes leader lines from a multileader entity.

Accessing the Command

command bar: *mleaderedit*
menu bar: *Dimension | Edit Multileader - Multileader Add Leaders - Multileader Remove Leaders*
toolbar: *Dimensions |*

: ***mleaderedit***
Prompts you in the command bar:

Select a multileader: **(Select an mleader entity.)**

Specify leader arrowhead location or [Remove leaders]: **(Pick a point.)**

Specify leader arrowhead location or [Remove leaders]: **(Pick a point or right click to stop adding leader lines.)**

Command Options

Option	Icon	Description
Add leaders		Allows to add leaders to the selected mleader entity. Prompts you: Select a multileader: Select a multileader entity. Specify leader arrowhead location or [Remove leaders]: Specify a point, then keep specifying points to add more leaders or right click to stop.
Remove leaders		Allows to remove leaders from the selected mleader entity. Prompts you: Select a multileader: Select a multileader entity. Select leaders to remove or [Add leaders]: Select a leader, then right click to remove the leader.

Procedures

Adding vertices to a leader line of a multileader
Removing vertices from a leader line of a multileader
Adding leader lines to a multileader
Removing leader lines from a multileader

Related Commands

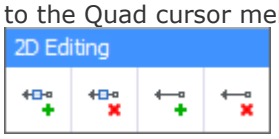
BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
MLeader - creates multileader entities using the current multileader style.
MLeaderEditExt - adds and removes leader lines, adds and removes vertices from a multileader entity.
MLeaderStyle - Creates and manages multileader styles in the Drawing Explorer dialog box.

MLeadereEditExt

Adds and removes leader lines, adds and removes vertices from a multileader entity.

Accessing the Command

With the Quad cursor menu active, pause the cursor over a multleader line, then move the cursor to the Quad cursor menu and expand the *2D Editing* command group.



Icon	Tool name	Description
	Add vertex	Adds a vertex to a multileader line.
	Remove vertex	Removes a vertex from a multileader
	Add Leader	Adds an leader line to a multileader.
	Remove Leader	Removes a leader line from a multileader.

Procedures

Adding vertices to a leader line of a multileader
Removing vertices from a leader line of a multileader
Adding leader lines to a multileader
Removing leader lines from a multileader

Related Commands

MLeaderEdit - Adds leader lines to and removes leader lines from a multileader entity.

MLeaderStyle

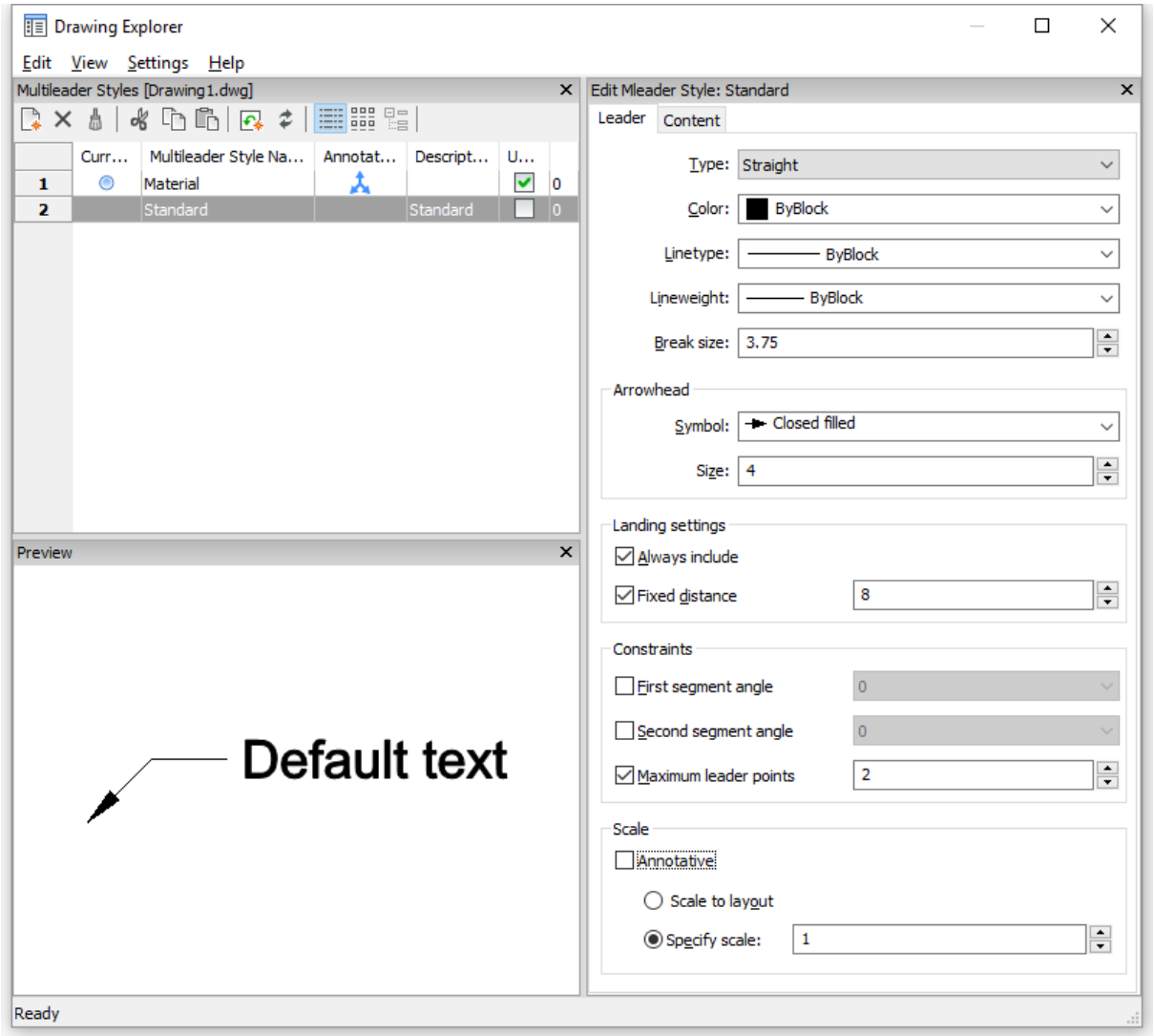
Creates and manages multileader styles through the Drawing Explorer.

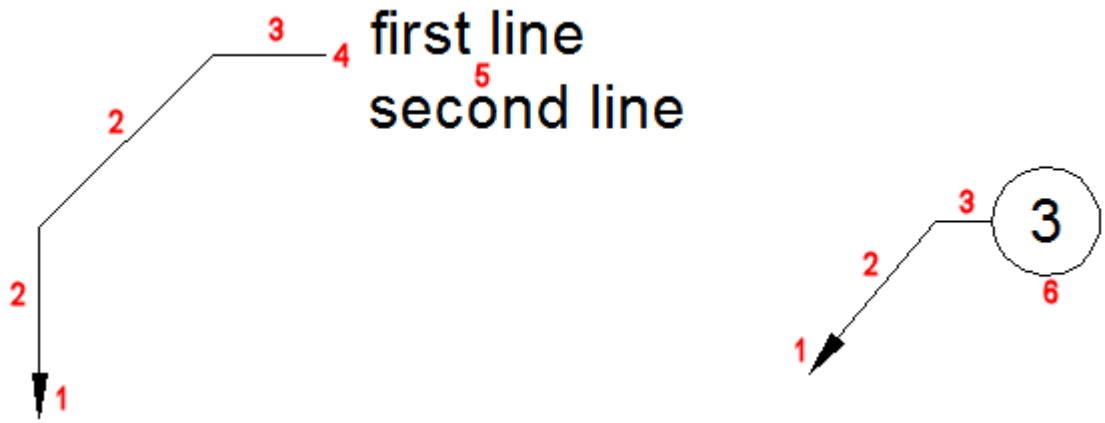
Accessing the Command

command bar: *mleaderstyle*
menu bar: *Tools | Drawing Explorer | Multileader Styles...*

: ***mleaderstyle***

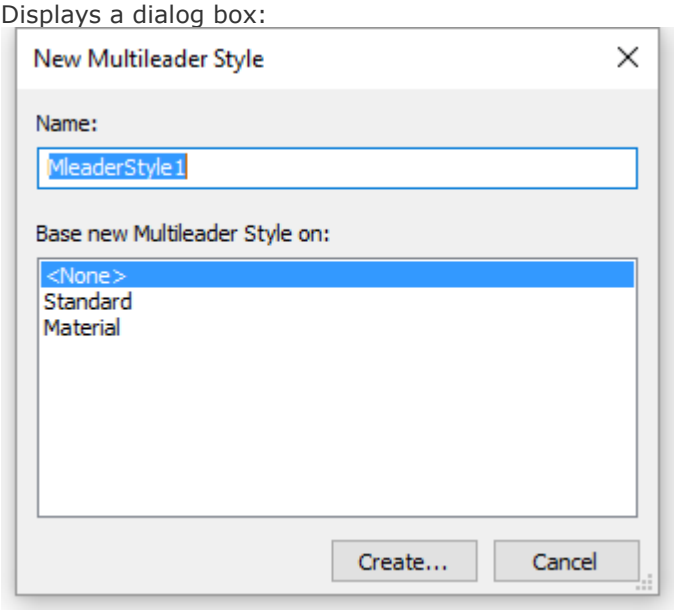
Displays a dialog box:





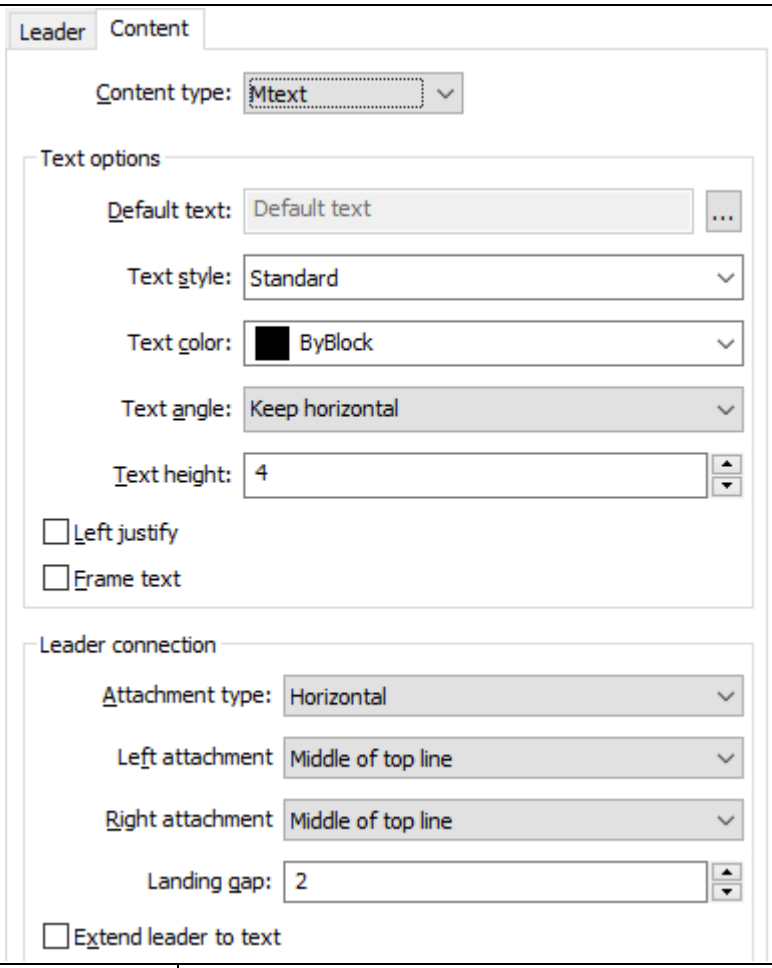
1. Arrowhead
2. Segment
3. Landing
4. Landing gap
5. Mtext
6. Block

Command Options

Option	Description
New	Creates a new multileader style. Displays a dialog box:  1. Type a name in the *Name* field. 2. (option) Select an existing style to create the new style as a copy of the existing style. 3. Click the *Create* button.
Delete	Deletes the selected multileader style. You cannot delete the *Standard* style or a style that is used in the drawing.

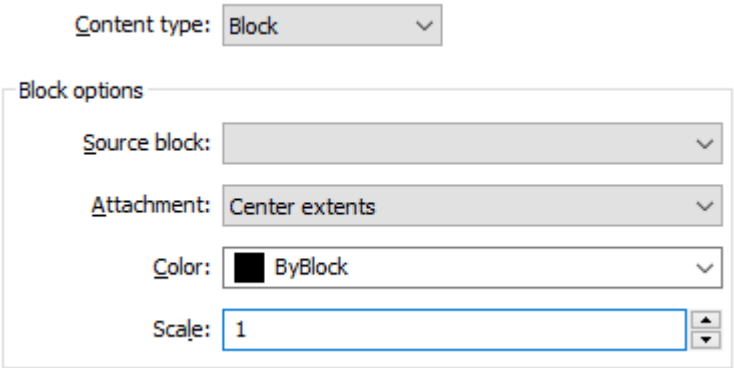
Leader Properties	
Type	The leader type can be either *Straight*, *Spline* or *None*.
Color	Sets the leader and arrowhead color.
Linetype	Sets the leader linetype
Lineweight	Sets the leader lineweight.
Break Size	Sets the break size used for the DIMBREAK command when the multileader is selected. (Not implemented yet)
Arrowhead	Determines the appearance of the leader arrowhead: • Symbol: select an arrowhead symbol from the list button. • Size: type a value to define arrowhead size in drawing units.
Landing Settings	Controls the creation and size of the leader landing: • Always include: Check the option to create a horizontal landing segment. • Size: Defines the landing length.
Constraints	Defines the constraints of the mleader entity. • First segment angle: Constrains the first leader segment. The angle of the first leader segment is constrained to of this value or a multiple of this. • Second segment angle: Constrains the second leader segment. The angle of the second leader segment is constrained to of this value or a multiple of this. • Maximum leader points: Specifies the maximum number of leader vertices.
Scale	Controls the scaling of the multileader entity. • Annotative: Sets the Annotative property of the multileader entity on. • Scale to layout: Applies a scaling factor with respect to the scaling in model space and paper space viewports. • Specify scale: Specifies the multileader scale.
Content	
Content type	Sets the content type of the mleader entity. • Mtext: Adds an Mtext entity. • Block: Adds a block. • None: Creates a leader only.

Mtext



Text options	Determines the appearance of the Mtext: • Default text: Specifies the default text. Click the ... button to launch the Mtext editor and type the default text. • Text style: Click the field to select a text style. • Text color: Choose a color. • Text angle: Sets the rotation angle of the Mtext. • Text height: Sets the text height. • Left justify: Forces the Mtext to always be left justified. • Frame text: Adds a box around the Mtext.

Leader connection	Specifies how the leader and the Mtext are connected.
	• Attachment type: Choose either Horizontal or Vertical.
	• Horizontal:
	  • Left attachment: Specifies the attachment of the landing line when the Mtext is to the right of the leader.
	  Choose an option in the list.
	  • Right attachment: Specifies the attachment of the landing line when the Mtext is to the left of the leader.
	  Choose an option in the list.
	• Vertical:

	• Top attachment: Specifies the attachment of the leader if the Mtext is below the leader. • Bottom attachment: Specifies the attachment of the leader if the Mtext is above the leader. • Landing gap: Defines the distance between the landing line and the Mtext or the space between the Mtext and the text frame. • Extend leader to text: If checked, extends the landing line to end at the edge of the text line where the leader is attached, not at the edge of the multiline text box. The length of the multiline text box is determined by the length of the longest line of text, not the length of the bounding box. Leader extension applies to horizontal leader attachment only.
Block	
	
Source block:	Specifies the block to be used.
Attachment:	Sets the attachment of the block to the leader. The options are: • Center Extents • Insertion point
Color:	Specifies te color of the block, if the entity color of the entities in the block is set to ByBlock.
Scale:	Sets the scaling of the source block.

Procedures

To create a multileader
Adding vertices to a leader line of a multileader
Removing vertices from a leader line of a multileader
Adding leader lines to a multileader
Removing leader lines from a multileader

Related Commands

MLeader - creates multileader entities using the current multileader style.
MLeaderEdit - adds leader lines to and removes leader lines from a multileader entity.
Explorer - opens the Drawing Explorer dialog box.

BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
Dimleader - places leaders with text, blocks, entities, or tolerances.
Leader - places leaders with text, blocks, entities, or tolerances.
QLeader - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

MLine

Draws multilines.

Accessing the Command

command bar: *mline*
aliases: *ml*
menu bar: *Draw | Multiline*

: ***mline***
Prompts you in the command bar:

Start of line or [Justification/Scale/STyle/Follow] <Last point>: (Pick a point in the drawing, or enter x,y,z coordinates, or choose an option.)

Set end point or [Angle/Length/Undo]: (Pick a point, or choose an option.)

Set end point or [Angle/Length/Follow/Undo]: (Pick a point, or choose an option.)

Set end point or [Angle/Length/Follow/Close/Undo]: (Pick more points, or choose an option, or press Enter to stop.)

The current multiline style is used to draw multilines. You can set the current multiline style using the MStyle command.

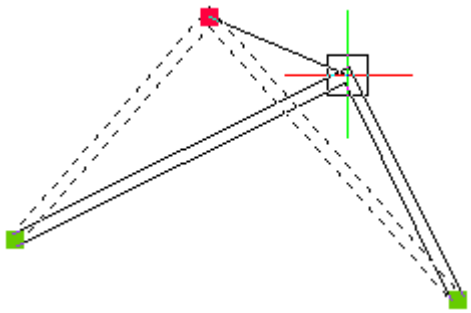
Command Options

Option	Description
Start of line	Specifies the starting point of the first multiline segment.
ENTER to use last point	Press Enter to use the most recently specified point.
Justification	Sets the CMLJUST system variable. Allows to specify the justification of the multiline with respect to the specified points in the drawing. Prompts you: Enter MLine justification type [Top/Zero/Bottom] <Top>: • Top: The top line of the multiline passes through the specified points. • Zero: The specified points define the zero-offset of the multiline, which is the center line if the multiline is defined symmetrically. • Bottom: The bottom line of the multiline passes through the specified points.
Scale	Sets the CMLSCALE system variable. Allows to specify the scaling of the multiline. Prompts you: Enter MLine scale <1.00>: Type the scale factor.
Style	Sets the CMLSTYLE system variable. Allows to choose a style for the multiline. Prompts you: Enter MLine style name [? to list available styles] - Type the name of a saved multiline style or type ? to see a list of available styles.
Follow	Uses the endpoint of the previously drawn multiline, polyline, line or arc as the start point for the next segment. The segment is drawn at the same angle of the previous line, multiline or polyline segment or tangent in case of an arc or polyline arc segment.
End point	Specifies the endpoint of the currently drawn segment.
Angle	Draws the segment at a user-specified angle and distance; prompts you: Angle of line - specify the angle (measured counter-clockwise from the positive x-axis).

	Length of line - specify the length of the segment.
Length	Draws the segment at a user-specified distance and angle; prompts you: Length of line - specify the length of the segment. Angle of line - specify the angle (measured counterclockwise from the positive x-axis).
Undo	Undoes (erases) the last segment.
Close	Closes the "polygon" by drawing a segment between the current endpoint and the first start point.

Grips Editing

1. Select the multiline. Notice that it has two or more grips: at the start and endpoint and at each vertex.
2. Drag a grip to stretch the multiline segment(s).



Related Commands

MStyle - creates and edits multiline styles.
Line - draws line segments.
PLine - draws connected lines and arcs.

MLStyle

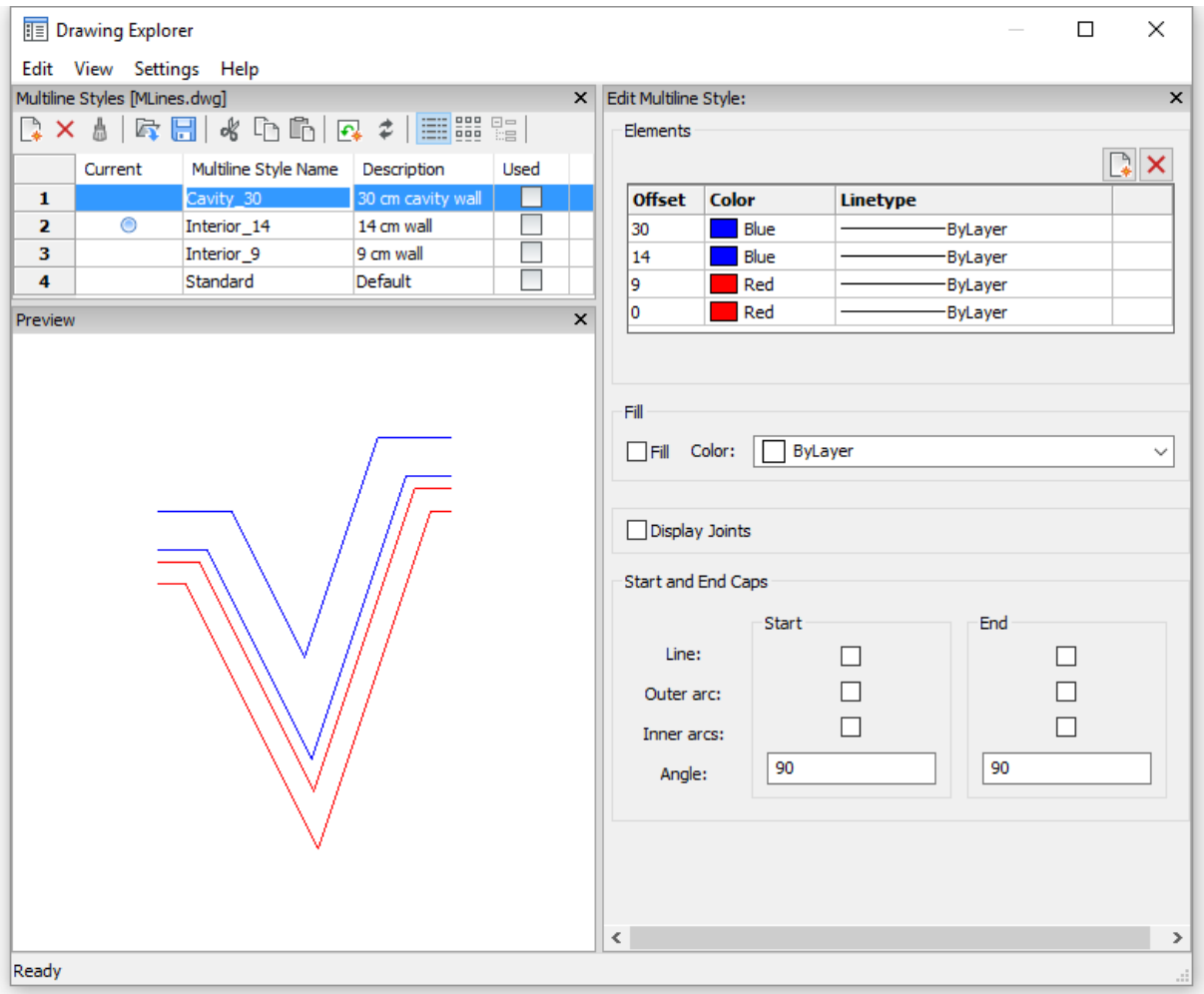
Creates and edits multiline styles.

Accessing the Command

command bar: *mlstyle*
menu bar: *Tools | Drawing Explorer | Multiline Styles...*

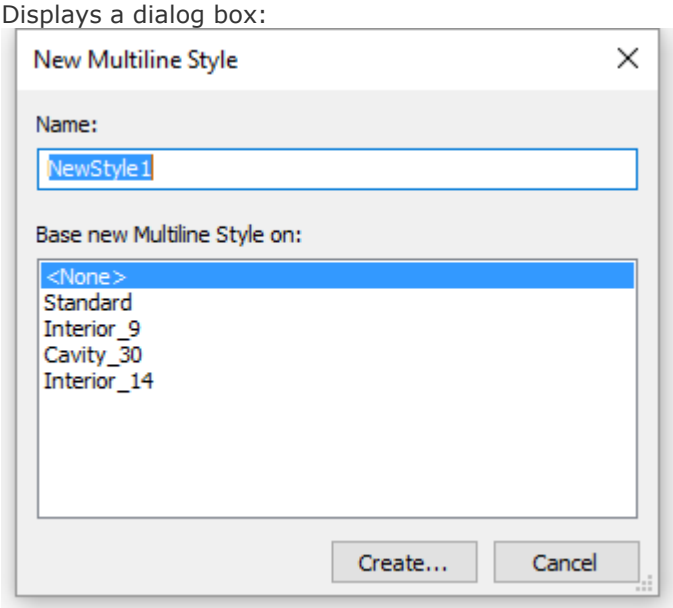
: ***mstyle***

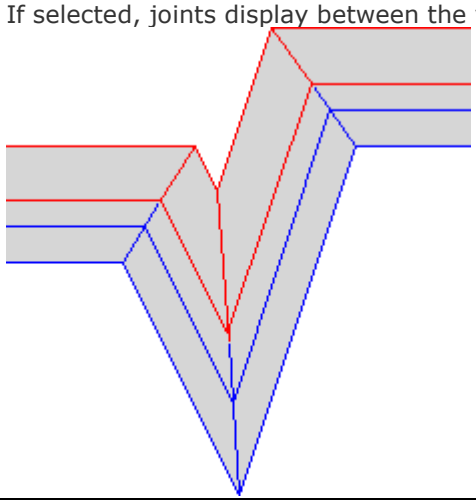
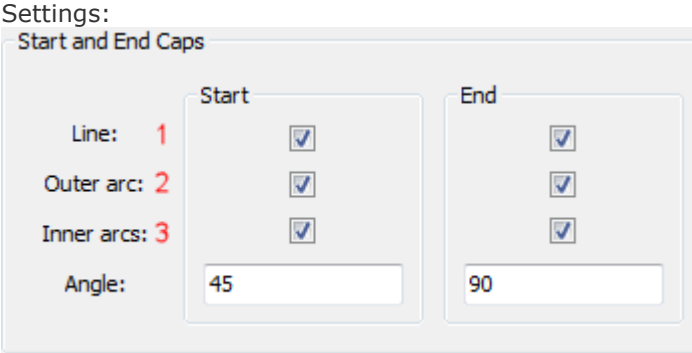
Displays the Drawing Explorer - Multiline Styles dialog box:

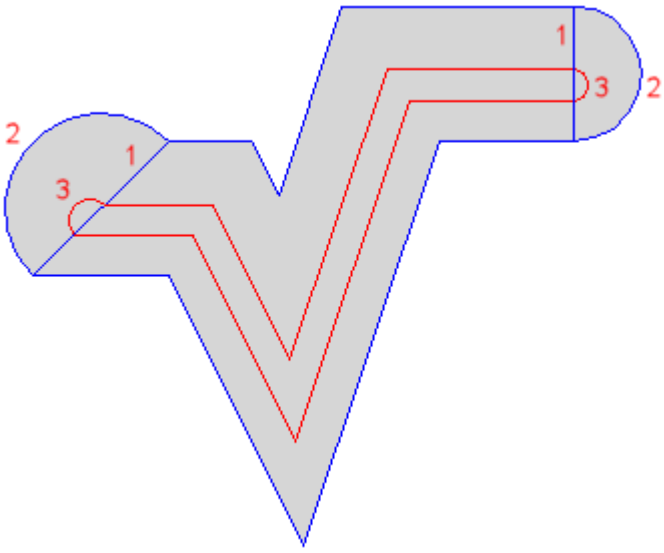


1. Details
2. Editor
3. Preview

Command Options

Multiline Details	
Option	**Description**
New	Creates a new multiline definition. Displays a dialog box:  Type a name in the *Name* field. Spaces are not accepted in multiline style names. (option) Select an existing multiline style, to be used as a template for the new style. Click the *Create* button. The New Multiline Style dialog closes. The new style is added in the *Details* pane. (option) Type a description in the *Description* field.
Delete	Deletes the selected style(s). Press and hold the Ctrl key to select multiple styles. Used styles cannot be deleted.
Purge	Purges the selected style(s).
Load from mln file	Loads multiline styles from a multiline (*.mln) file. Displays a file dialog box.
Save to mln file	Saves all multiline styles in the drawing to a multiline (*.mln) file. Displays a file dialog box.
Detail view	Switches to *Detail* view, when in *Icon* view.
Icon view	Switches to *Icon* view, when in *Detail* view.
	The current multline style is marked. Click the tile in front of a style to make it current.
Multiline Editor	
Add	Adds a new line.

	• Specify the offset from the reference line (0). A positive offset creates a line at the left hand side of the reference line, a negative offset creates a line at the right hand side. • Specify the line color. • Specify the linetype. Note: The number of lines and the offsets of used styles cannot be changed.
Delete	Deletes the selected line(s). Press and hold the Ctrl key to select multiple lines. A multiline style contains at least one line.
Fill	If selected, a solid fill is created between the outer lines.
Display joints	If selected, joints display between the various multiline segments. 
Start and End caps	Closes the end segments of open multilines. Settings:  Preview:

Related Commands

Mline - creates multilines.
Explorer - opens the Drawing Explorer dialog box.

ModelerProperties

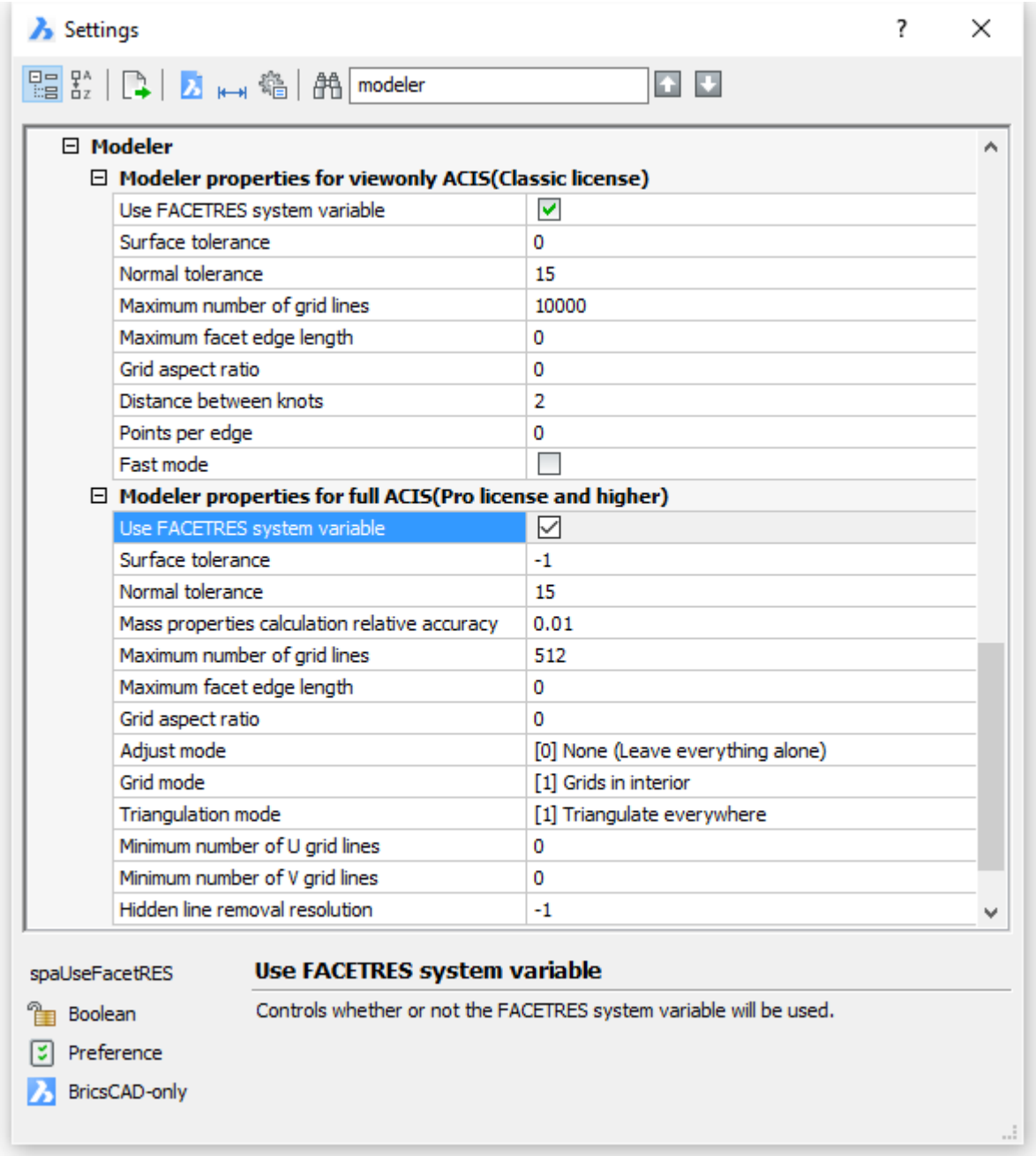
Controls the various settings of the Acis modeler through the *Settings* dialog box.

Accessing the Command

command bar: *modelerproperties*

: ***modelerproperties***

Displays the Modeler properties for edit ACIS user preferences in the Settings dialog box:



Command Options

Option	Description
Use FACETRES system vairable	Sets the *spaUseFacetRes* user preference. Controls whether the FACETRES system variable is used. 0 = do not use Facetres; 1 = use Facetres
Surface tolerance	Sets the *spaSurfaceTol* user preference. The surface tolerance specifies the maximum distance between a facet edge and the true surface. The proper value is dependent on the model size.
Normal tolerance	Sets the *spaNormalTol* user preference. The normal tolerance specifies the maximum normal deviation allowed between two normals on two adjacent nodes, expressed in degrees. The proper value is usually independent of the model size.
Mass Properties	Sets the *MassPropAccuracy* user preference. Specifies the relative accuracy required for mass properties calculations. Since the specified accuracy is relative, the calculation result depends on the magnitude of the calculated value. A proper value lies between 1.e-2 and 1.e-12.
Maximum number of grid lines	Sets the *spaMaxNumGridLines* user preference. Specifies the maximum number of grid subdivisions. This prevents the facet data of a face from getting too big.
Maximum facet edge length	Sets the *spaMaxFacetEdgeLength* user preference. Specifies the maximum length of a side of a cell in object space. Since a facet cannot be larger than the cell, this determines the maximum size of a facet.
Grid aspect ratio	Sets the *spaGridAspectRatio* user preference. Specifies the approximate aspect ration of each cell in the grid. If the value is close to 1, then the cell is close to a square. This does not guarantee the aspect ration of the facet, which may consist of only part of a cell.
Adjust mode	Sets the *spaAdjustMode* user preference. The adjust mode is used for triangle smoothing.
Grid mode	Sets the *spaGridMode* user preference. Controls how gridding is used in the mesh process.
Triangulation mode	Sets the *spaTriangMode* user preference. Triangulation mode identifies which portion of the mesh is to be triangulated.
Minimum number of U grid lines	Sets the *spaMinUGridLines* user preference. Controls the minimum number of U grid lines. A least this number of grid lines are generated in U direction.
Minimum number of V grid lines	Sets the *spaVGridLines* user preference. A least this number of grid lines are generated in V direction.

Hidden line removal resolution	Sets the *AcisHlrResolution* user preference. Specifies the accuracy to be used for mass properties calculations. This accuracy is relative: the accuracy of the result depends on the magnitude of the calculated value. Acceptable value range: 0.01 - 0.000000000001 (1.e-2 to 1.e-12). Specifies the smallest distance taken into consideration during Hidden Line Removal calculation. A negative value means that auto-calibration based on the size of the model will be used. This is highly recommended. When dealing with objects that have very small sizes the value can be set to 0.001 or smaller. The value of the *AcisHlrResolution* user preference can also be set in the FLATSHOT dialog box.

Related Commands

-ModelerProperties - controls the various settings of the Acis modeler through the command bar.

-ModelerProperties

Controls the various settings of the Acis modeler through the command bar.

Accessing the Command

command bar: *-modelerproperties*

: ***-modelerproperties***
Prompts you in the command bar:
[view Properties/USe facetres/Surface tolerance/Normal tolerance/Max grid lines/Edge length/grid aspect Ratio/Adjust mode/Grid mode/Triangulation mode/min U grid lines/min V grid lines/mass properties aCcuracy/eXit]<eXit>: *(select an option)*

Command Options

Option	Description
View Properties	Prompts for the View Acis properties.
Use Facetres	Sets the *spaUseFacetRes* user preference. Controls whether the FACETRES system variable is used. 0 = do not use Facetres; 1 = use Facetres
Surface Tolerance	Sets the *spaSurfaceTol* user preference. The surface tolerance specifies the maximum distance between a facet edge and the true surface. The proper value is dependent on the model size.
Normal Tolerance	Sets the *spaNormalTol* user preference. The normal tolerance specifies the maximum normal deviation allowed between two normals on two adjacent nodes, expressed in degrees. The proper value is usually independent of the model size.
Maximum Grid Lines	Sets the *spaMaxNumGridLines* user preference. Specifies the maximum number of grid subdivisions. This prevents the facet data of a face from getting too big.
Edge Length	Sets the *spaMaxFacetEdgeLength* user preference. Specifies the maximum length of a side of a cell in object space. Since a facet cannot be larger than the cell, this determines the maximum size of a facet.
Grid Aspect Ratio	Sets the *spaGridAspectRatio* user preference. Specifies the approximate aspect ration of each cell in the grid. If the value is close to 1, then the cell is close to a square. This does not guarantee the aspect ration of the facet, which may consist of only part of a cell.
Adjust Mode	Sets the *spaAdjustMode* user preference. The adjust mode is used for triangle smoothing.
Grid Mode	Sets the *spaGridMode* user preference. Controls how gridding is used in the mesh process.
Triangulation Mode	Sets the *spaTriangMode* user preference.

	Triangulation mode identifies which portion of the mesh is to be triangulated.
Minimum U Grid Lines	Sets the *spaMinUGridLines* user preference. Controls the minimum number of U grid lines. A least this number of grid lines are generated in U direction.
Minimum V Grid Lines	Sets the *spaVGridLines* user preference. A least this number of grid lines are generated in V direction.
Mass Properties Accuracy	Sets the *MassPropAccuracy* user preference. Specifies the relative accuracy required for mass properties calculations. Since the specified accuracy is relative, the calculation result depends on the magnitude of the calculated value. A proper value lies between 1.e-2 and 1.e-12.

Related Commands

ModelerProperties - controls the various settings of the Acis modeler through the *Settings* dialog box.

Move

Moves entities.

Accessing the Command

command bar: *move*
alias: *m*
menu bar: *Modify | Move*
toolbar: *Modify |*

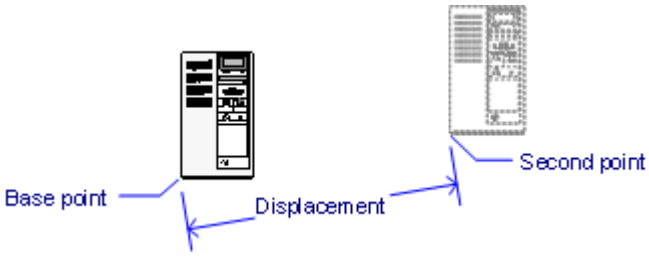
: ***move***
Prompts you in the command bar:

Select entities to move: (Choose one or more entities.)
Entities in set: 1

Select entities to move: (Choose more entities, or press Enter to continue.)

Enter base point [Displacement] <Displacement>: (Pick a point, enter x,y coordinates, or type D.)

Enter second point <Use base point as displacement>: (Pick another point, or press Enter.)



Command Options

Option	Description
Select entities to move	Specifies the entities to be moved.
Enter base point	Specifies the starting point. Pick a point in the drawing, or enter x,y coordinates. Use entity snap modes to ensure accuracy. To move entities horizontally or vertically, hold down the Shift key.
Enter second point	Specifies the destination for the entities. Pick a point in the drawing, or enter x,y coordinates.
Displacement	Specifies the displacement vector to move the entities; prompts you: **Enter displacement vector <100,200,0>** - Enter x,y,z coordinates to specify the displacement vector from the origin of the current UCS. The z-coordinate is optional. If not specified z = 0.
Use base point as displacement	Uses the x,y,z coordinates of the base point to define the displacement vector.

Procedures

Using the Move command
Moving entities using grips
Using Cut and Paste to move entities between drawings
Move a selection as a block
To move entities using grips editing

Related Commands

PEdit - moves the vertices of polylines.
CutClip - moves entities from the drawing to the Clipboard.

MoveEData

Moves extended entity data from one entity to others.

Accessing the Command

command bar: *moveedata*
transparent: *'moveedata*
menu bar: *Tools | Entity Data | Move Entity Data*
toolbar: *Entity Data |*

: ***moveedata***

Prompts you in the command bar:

Move Entity Data: Name of application or [? to list application names]: (Enter name of application, or type ?.)

Select entity from which to move data: (Choose one entity.)

Select entity to which to move data: (Choose another entity.)

Select entity from which to move data: (Choose another entity, or press Enter to exit the command.)

Command Options

Option	Description
Name of application	Specifies the name of the application that generated the extended entity data.
? to list application names	Lists the names of registered applications.
Select entity From which to move data	Selects the source entity from which to move edata. The entity must contain edata, which can be created with the EditEData command.
Select entity to which to move data	Specifies the target entity to receive the edata.

Related Commands

CopyEData - copies extended entity data to other entities.
DelEData - erases extended entity data from entities.
EditEData - creates and edits extended entity data.
ReAssocApp - associates applications with entities.

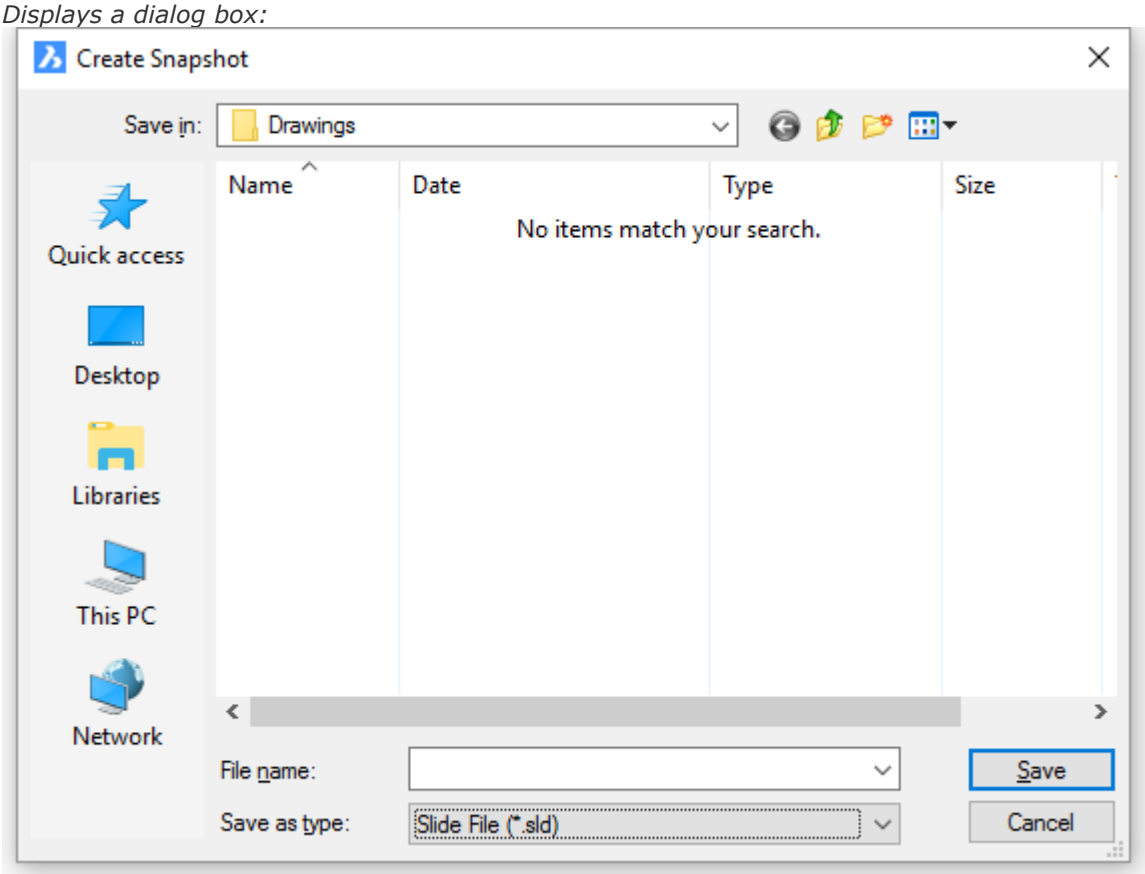
MSlide

Creates slides (snapshots) of the current drawing (short for "make slide").
Limited support on Linux: export to *Slide Files (*.sld)* only.

Accessing the Command

command bar: *mslide*
alias: *msnapshot*
menu bar: *Tools | Make Snapshot*
toolbar: *Tools |*

: ***mslide***
Displays a dialog box:



Enter a file name, choose a format, and then click Save.
To view slides, use the **VSlide** command.

Command Options

Option	Description
File name	Specifies the name of the slide (snapshot).
Save as type	Specifies the file format: • SLD - compatible with AutoCAD. • EMF - enhanced WMF format. • WMF - Windows meta format.
Save	Saves the slide file.

Related Commands

VSlide - views WMF, EMF, and SLD files.
BmpOut - exports drawings in BMP format.
Export - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats.
WmfOut - exports drawings in WMF, EMF, and SLD formats.

MSpace

Switch to a model space viewport in a layout tab.

Accessing the Command

command bar: *mspace*
shortcut: *Double-click inside the viewport*
alias: *ms*
status bar: *P:Layout*

: ***mspace***

(This command operates only when the layout tab is in paper space.)

Displays no prompt in the command bar.

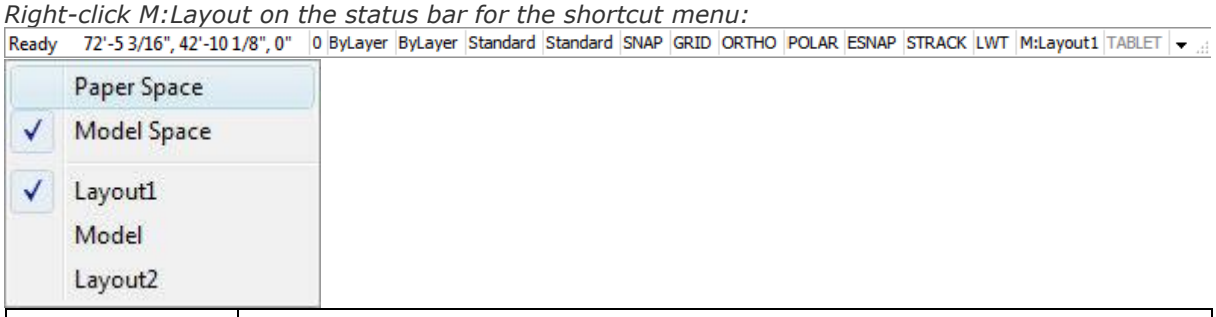
*To reverse the action of this command, use the **PSpace** command, or double-click outside the viewport border.*

Command Options

No options.

Status Bar Menu

Right-click M:Layout on the status bar for the shortcut menu:



Shortcut Menu	Description
Paper Space	Switches to paper space in the current layout.
Model Space	Switches to model space in the current layout.
Layout1	Switches the drawing editor to Layout1.
Model	Switches the drawing editor to Layout1.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Related Commands

ChSpace - Moves entities from paper space to model space and vice versa.
MView - creates viewports in paper space.
VpClip - clips viewports.
PSpace - switches to paper space.
VPorts - creates viewports in model space.
RedrawAll - redraws all viewports.
RegenAll - regenerates the drawing in all viewports.

Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Mtp

Allows to specify a point as the midpoint between two points whenever a command prompts to enter a point (transparent command).

Accessing the Command

transparent: *mtp*
alias: *m2p*
toolbar : *Entity Snaps* |

<running command prompt>: ***mtp***
Prompts you in the command bar:

First point of mid: (Identify the first reference point)

Second point of mid: (Identify the second reference point)

Command Options

None

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

MText

Places multi-line text in drawings through a dialog box.

Accessing the Command

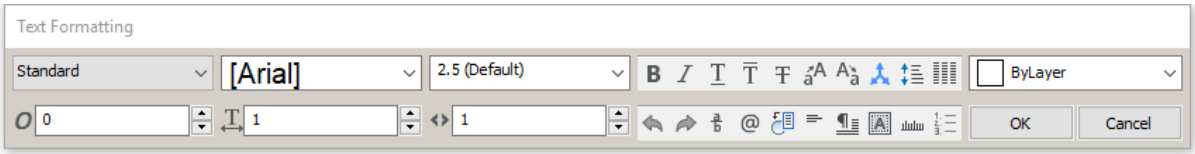
command bar: *mtext*
aliases: *t, mt*
menu bar: *Draw | Multiline Text*
toolbar: *Draw |*

: ***mtext***
Prompts you in the command bar:

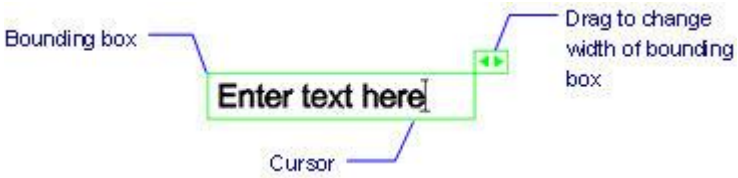
Multiline Text: First corner for block of text: (Pick a point, or enter x,y coordinates.)

Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text Height/Direction/Width]: (Pick another corner, or enter an option.)

Displays a Text Formatting toolbar and text entry field:

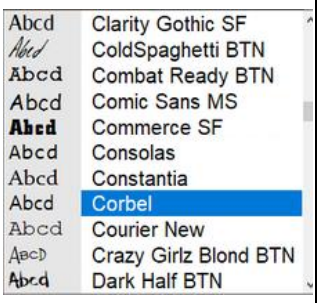
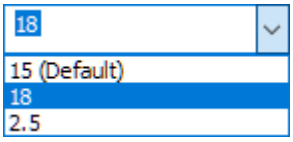


Enter text, and then click OK.



Command Options

Option	Description
Style Standard ASHADE DYN_DIM Roman Standard	Specifies the style. Choose a style name from the droplist. See Style command.

Font  	Specifies the font. Choose a font name from the droplist; this overrides the font specified by the style. BricsCAD can use any font installed on your computer. To apply a font to just one or two words, highlight them, and then choose the font from the droplist, as illustrated below: To apply a font to just one or two words, **highlight them**, and then choose the from the droplist.
Height  	Specifies the text height. Choose a preset height from the droplist, or enter a value. To change the height of a few words, highlight them, and then specify the height.
B Bold	Toggles boldfacing of text. SHX fonts cannot be boldfaced. To apply boldfacing: 1. Select some or all of the text, and then click the B button. 2. Repeat the procedure to remove boldfacing.
I Italic	Toggles italicization of text. SHX fonts cannot be italicized. To apply *italics*: 1. Select some or all of the text, and then click the I button. 2. Repeat the procedure to remove italics.
T Underline	Toggles underlining of text. To apply underlining: 1. Select some or all of the text, and then click the U button. 2. Repeat the procedure to remove underlines.
T Overline	Toggles overlining of text. To apply overlining: 1. Select some or all of the text, and then click the O button. 2. Repeat the procedure to remove overlines.
T Strikethrough	Toggles the strikethrough of the text.
aA Upper case	Sets selected text in upper case.
Aa Lower case	Sets selected text in lower case.

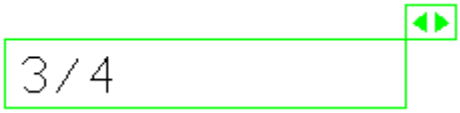
Annotative	Toggles the annotative property of the MText. • If the annotative property is currently off, press the Annotative button to apply the current annotative scale. • If the annotative property is currently on, press the Annotative button to switch the annotative property off.
Line spacing	Sets the line spacing. ✓ 1.0x 1.5x 2.0x 2.5x Clear line space
Color ByLayer	Specifies the color of the text. 1. Select some or all of the text. 2. Click the Color button, and then choose a color from the droplist.
Oblique Angle 0	Specifies the angle by which to slant the text. • Enter an angle, or click the buttons to change the value. • Less than 0 degrees - characters slant backwards. • 0 - characters are upright. • More than 0 degrees - characters slant forwards.
Width Factor 1	Specifies the width of characters. • Smaller than 1 - characters are narrow. • 1 - characters have normal width. • Greater than 1 - characters are wide.
Character Spacing 1	Specifies the spacing between characters. • Smaller than 1 - characters spacing is tight. • 1 - characters have normal spacing. • Greater than 1 - characters spacing is loose.
Undo	Undoes the last format applied.
Redo	Redoes the last undo.

Stack Fractions	Stacks text containing special characters: / # or ^. • / stacks vertically (separated by a horizontal line)

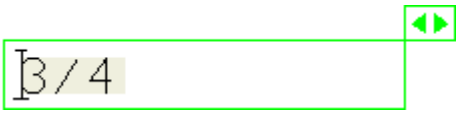
- # stacks diagonally (separated by a diagonal line)
- ^ stacks vertically not separated by a line (tolerance stack)

To stack fractions:

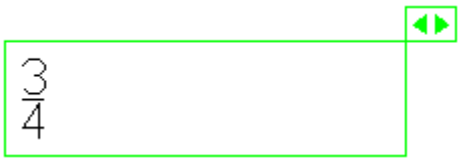
1. Enter fraction text, such as 3/4.



2. Select the fraction text; e.g. doubl click.



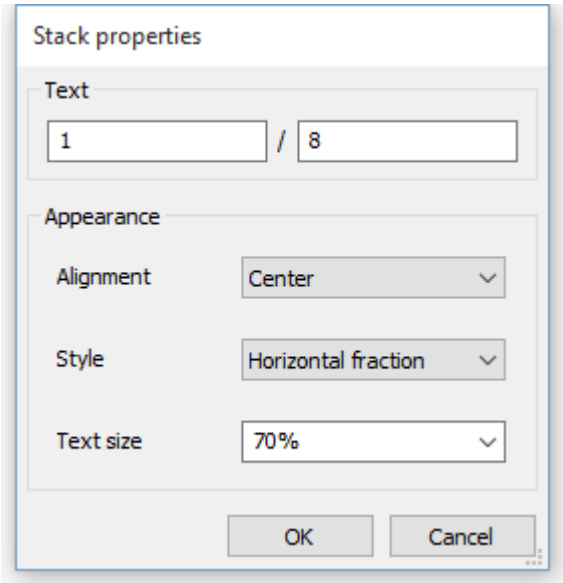
3. Click the *Stack Fraction* button ( ).



4. Repeat steps 2 and 3 to unstack the fractions.

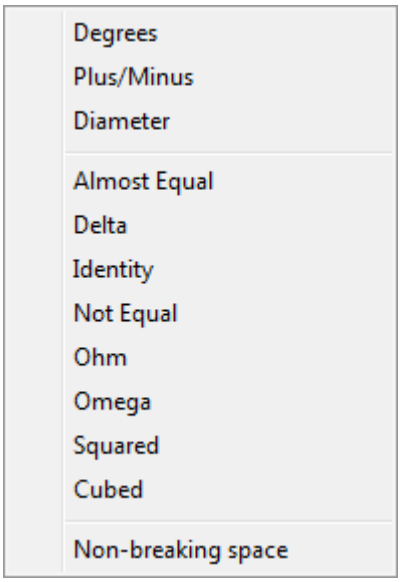
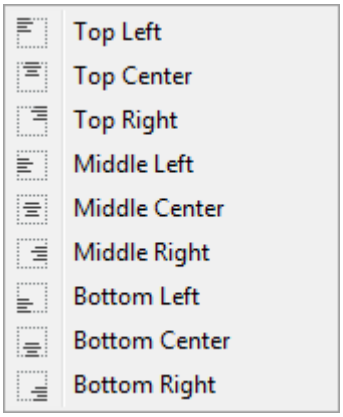
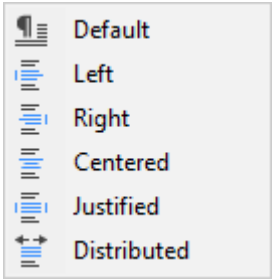
To edit the stack properties:

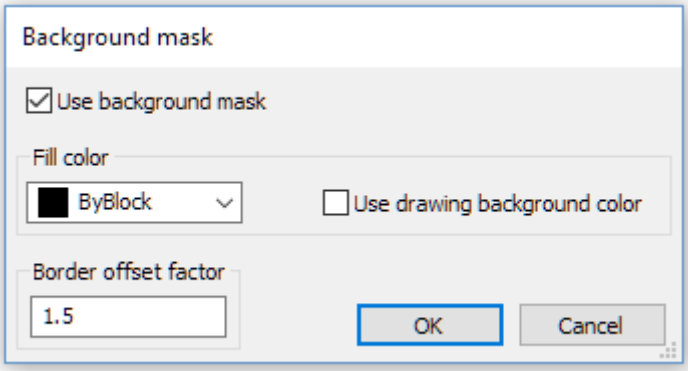
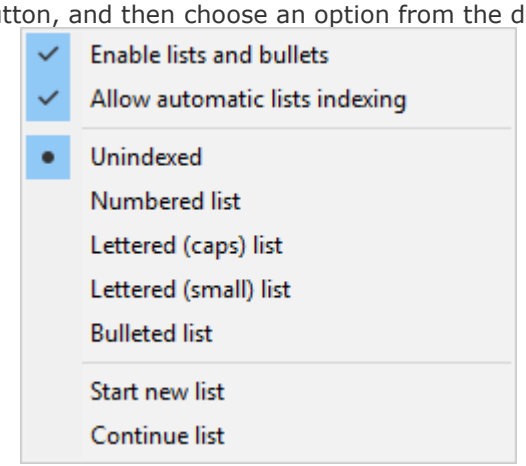
1. Double click the stacked text.
   The *Stack Properties* dialog displays:



- Text: type a value
- Alignment: select an option.
- Style: Select an option
- Text size: Type a value or select one in the list.

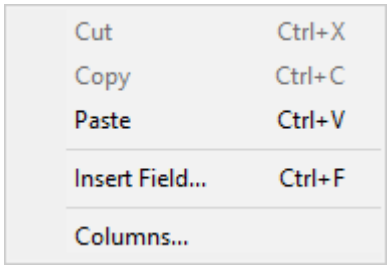
2. Click the *OK* button to apply the changes.

Insert Symbol	Inserts special symbols. Click the button, and then choose a symbol.  You can enter the symbols directly by typing their codes, such as %%d for the degree symbol.
Field	Inserts a field.
Text justification	Specifies the text justification (alignment) within the bounding box. Click the button, and then choose an alignment from the droplist.  The icon on the button indicates the current justification mode.
Paragraph	Specifies the text alignment of a paragraph. Click the button, and then choose an alignment from the droplist. 

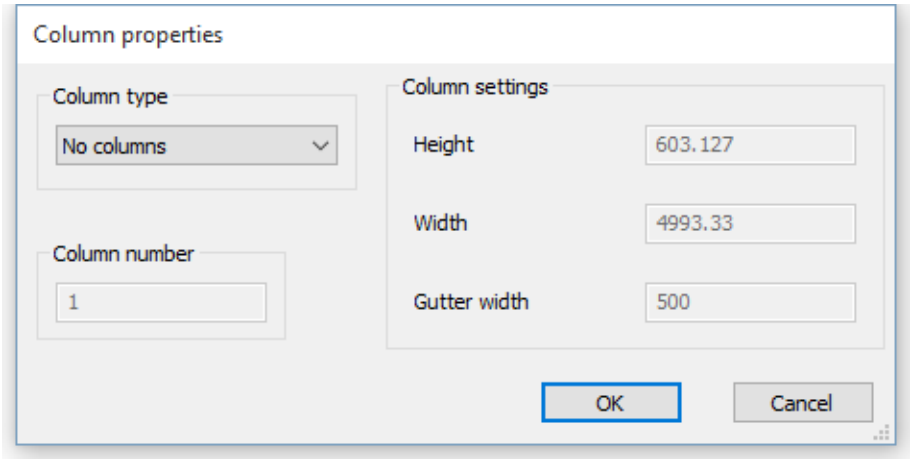
Background Mask	Sets the background mask. 
Ruler	Toggles the display of the ruler. 1. First line indent marker 2. Second line indent marker 3. Right indent marker 4. Tab type
Lists	Creates numbered and bulleted lists. Click the button, and then choose an option from the droplist.  • Enable lists and bullets: Enables the creation of lists. • Allow automatic lists indexing: If selected, Enter adds a new list item.

Columns

To divide the text in columns:
Right click and choose *Columns...* in the context menu.

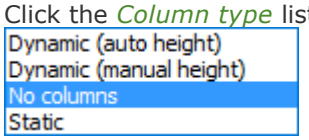


The Column Properties dialog box displays:

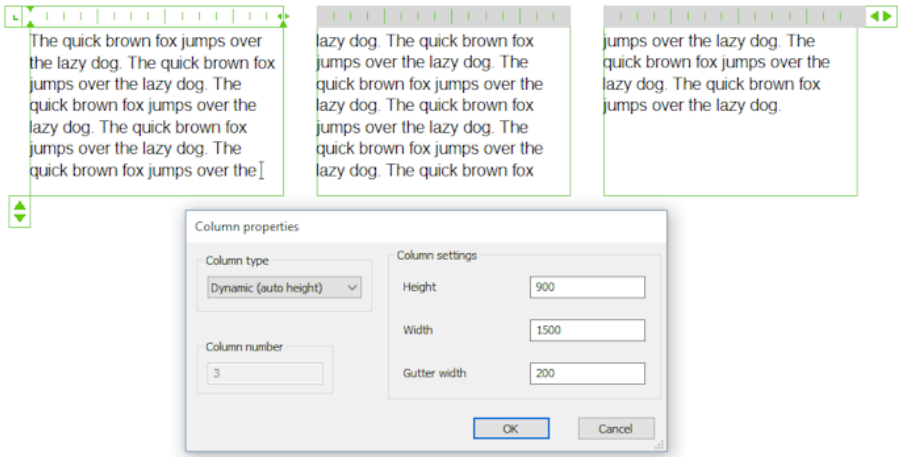


- Column type: defines how the number of columns is controlled.
- Column number: number of columns
- Height: maximum height of the columns.
- Width: width of each column.
- Gutter width: space between the columns.

Click the *Column type* list button and select a column type in the list:

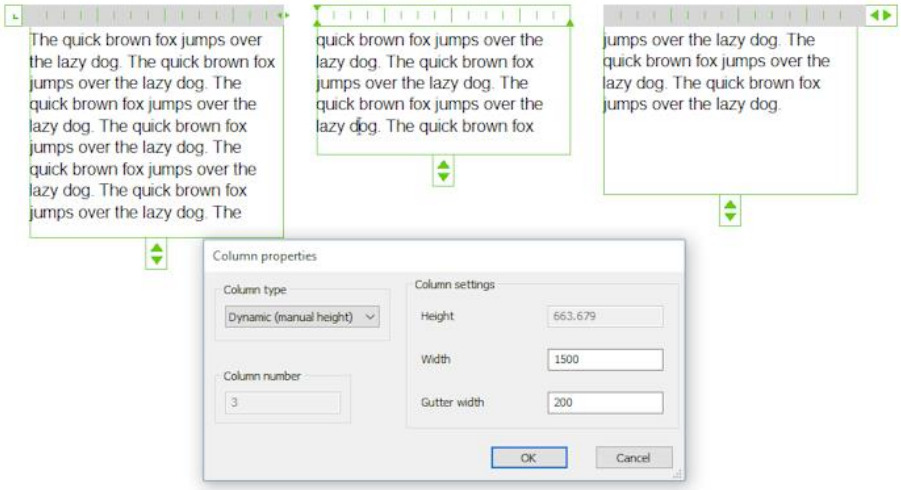


- Dynamic (auto height):



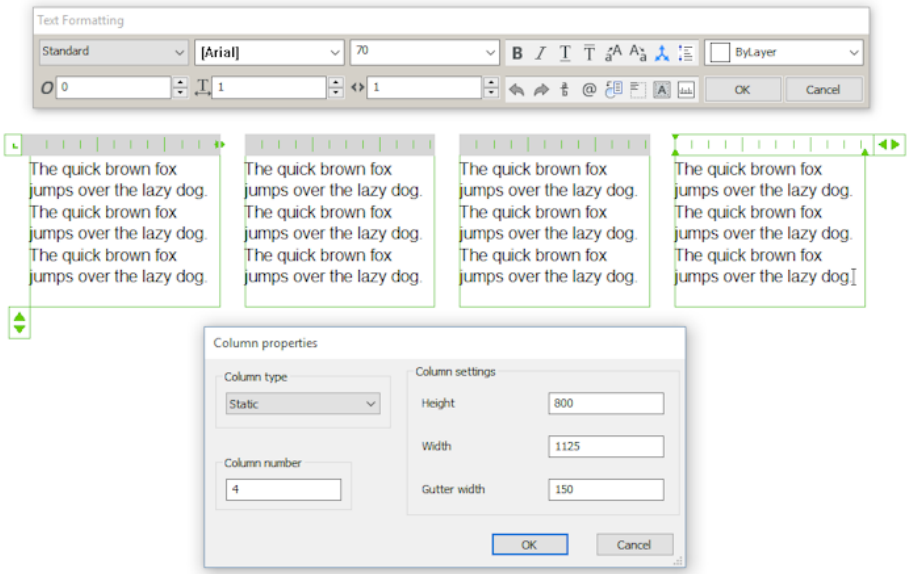
Type a value in the *Height*, *Width* and *Gutter width* fields. The number of columns is calculated automatically
Optionally, adjust the column height by dragging the control button.

- Dynamic (manual height):



Type a value in the *Width* and *Gutter width* fields. The number of columns is calculated automatically.
The *Height* can be controlled by dragging the control button for each column separately.

- Static:



Type a value in the *Column number*, *Height*, *Width* and *Gutter width* fields.
Optionally, adjust the column height by dragging the control button.

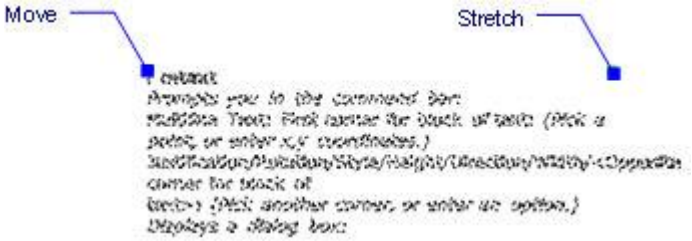
Procedures

To create single line text
To create multiline text
Including special text characters and symbols
To add a background mask
General procedure to create fields
Defining text system variables

Grips Editing

Mtext bounding boxes can be edited directly through grips:

1. Select the mtext. Notice that it has two grips.
2. Drag the grips to perform the following actions:
   - Drag the left grip to move the mtext.
   - Drag the right grip to stretch the width of the bounding box (make it wider or narrower; the text reflows automatically to accommodate the change in width).



Related Commands

-MText - places paragraph text.
Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
AttDef - defines attribute text.
DdEdit - edits text.
Field - inserts a field in a multiline text, table cell or attribute.
Updatefield - Updates fields in selected Mtexts.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
QText - toggles between text and rectangular representations.
Change - changes the position of text.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Find - finds and replaces text.
Properties - modifies the properties of text.

-MText

Places multi-line text in drawings through the command bar.

Accessing the Command

command bar: *-mtext*

: ***-mtext***
Prompts you in the command bar:

Multiline Text: First corner for block of text: (Pick a point, or enter x,y coordinates.)

Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text Height/Direction/Width]: (Pick another corner, or enter an option.)

Mtext: (Enter text.)

Mtext: (Enter more text, or press Enter to exit the command.)

Command Options

Option	Description
First corner for block of text	Specifies one corner of the rectangle that bounds the text block.
Opposite corner for block of text	Specifies the other corner.
Justification	Specifies the justification (horizontal alignment) of the text inside the bounding box; prompts you: Specify text block justification [Top-Left/Top-Center/Top-Right/MiddLe-left/Middle-Center/Middle-Right/Bottom-Left/Bottom-Center/Bottom-Right] <Top Left> - enter a justification mode.
Rotation	Specifies the rotation angle of the text block; prompts you: Rotation angle for block of text <0> - enter an angle. Positive angles rotate the text block counter-clockwise.
Style	Specifies the text style to use for the mtext; prompts you: **Text style to use or [? to list] <Standard>** - enter the name of a text style. Enter ? for a list of styles defined in the current drawing. See **Style** command.
Height	Specifies the height of the text; prompts you: Height of text <0.2000> - enter a number. This option is unavailable when the style specifies the text height .
Direction	Specifies the direction in which the bounding box is expanded; prompts you: Specify drawing direction [Left-to-right/Top-to-bottom/by-Style] <Left-to-Right>: **- enter an option:** • Left to right - text is placed at the left, and expands the bounding box to right. • Top to bottom - text is placed at the top, and expands the bottom of the bounding box. • By Style - returns to the previous prompt.
Width	Specifies the width of the bounding box; prompts you:

	Width of text box - enter a value. When set to 0, the box has "infinite" width.
Mtext	Specifies the text; prompts you: Mtext - specifies the text; to end the command, press Enter at this prompt.

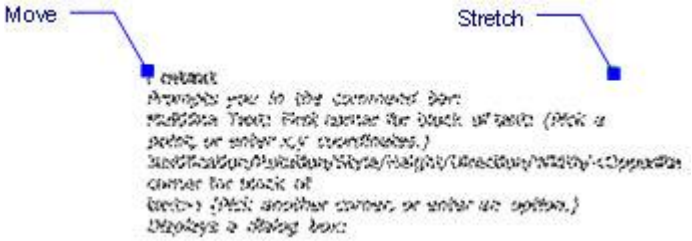
Procedures

To create single line text
To create multiline text
Including special text characters and symbols
To add a background mask
General procedure to create fields
Defining text system variables

Grips Editing

Mtext bounding boxes can be edited directly through grips:

1. Select the mtext. Notice that it has two grips.
2. Drag the grips to perform the following actions:
    - Drag one grip to move the mtext.
    - Drag the other grip to stretch the width of the bounding box (make it wider or narrower; the text reflows automatically to accommodate the change in width).



Related Commands

Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
Change - changes the position of text.
QText - toggles between text and rectangular representations.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Find - finds and replaces text.
Properties - modifies the properties of text.

Multiple

Repeats commands until you press Esc.

Accessing the Command

command bar: *multiple*

: ***multiple***
Prompts you in the command bar:
Enter command name to repeat: (Enter a command.)

The command is executed and then repeated until you press Esc.

Command Options

No options.

Related Commands

All.

MView

Creates one or more viewports in layout tabs (short for "make viewports").

Accessing the Command

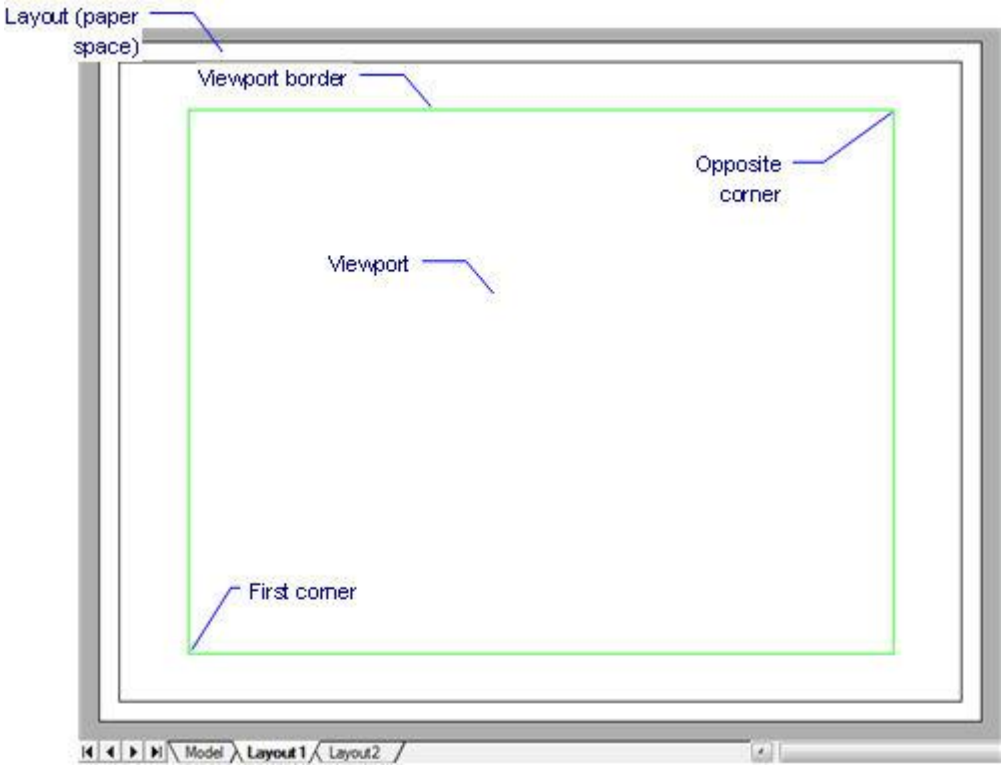
command bar: *mview*
alias: *mv*
menu bar: *View | PaperSpace Views*
toolbar: *View |*

: ***mview***

(This command runs only in layout tabs. To create viewports in model tab, use the **Vports** command.)

Prompts you in the command bar:
Viewports: Set first corner or [ON/OFf/Lock/Fit/create 2 viewports/create 3 viewports/create 4 viewports/Object/Polygonal]: *(Pick a point, or enter an option.)*

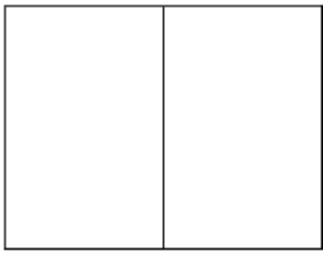
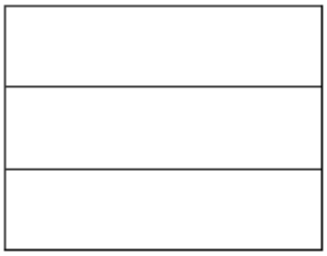
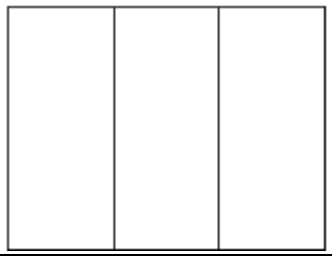
Opposite corner: (Pick another point.)

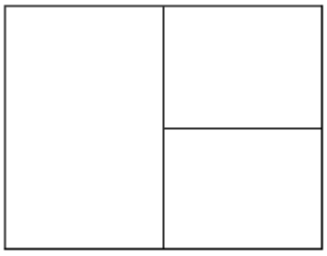
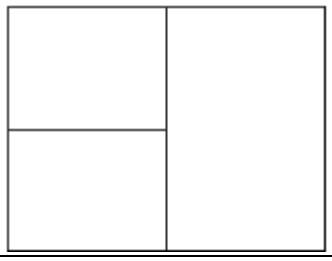
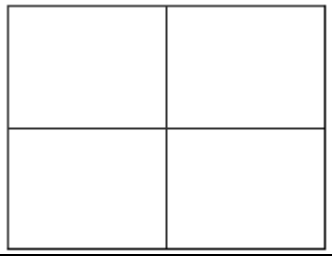


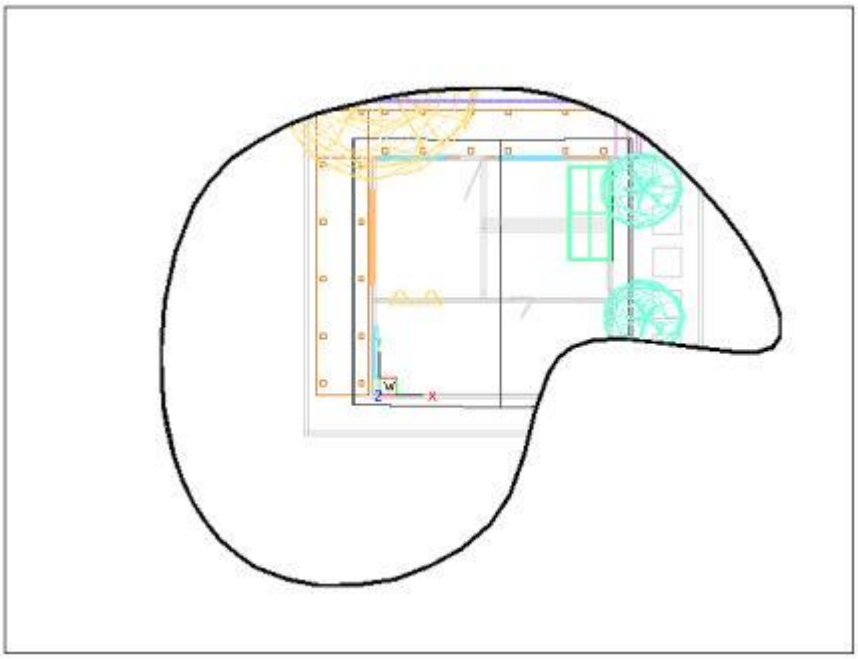
The viewport allows layouts to display entities drawn in model tab.

Command Options

Option	Description
First corner	Specifies one corner for a new, rectangular viewport. Pick a point, or enter x,y coordinates.
Opposite corner	Specifies the other corner for a new viewport.
ON	Turns on viewports that are turned off; prompts you: Select viewports to turn on - choose one or more viewports.
OFF	Turns off viewports; prompts you: Select viewports to turn off - choose one or more viewports. This option hides the content of the viewport; the viewport border remains visible. To hide the viewport border, place it on a separate layer, and then freeze the layer.
Lock	Locks the viewport's scale factor, relative to model space; prompts you: Viewport locking [ON/OFf] <On> - type ON or OFF. • ON - locks the viewport's scale factor. • OFF - unlocks the viewport's scale factor. Set the scale factor with the Zoom command's XP option.
Fit	Draws a rectangular viewport that fits the current layout.
2	Draws two rectangular viewports; prompts you: Two viewports [Horizontal/Vertical] <Vertical> - enter an option. **First corner of bounding rectangle or [Fit]** - Pick a point or choose the Fit option. Opposite corner - Pick a point.
Horizontal	Draws two horizontal viewports.

Vertical	Draws two vertical viewports. 
3	Draws three rectangular viewports; prompts you: **Three viewports [Horizontal/Vertical/Above/Below/Left/Right] <Right>** - enter an option. **First corner of bounding rectangle or [Fit]** - Pick a point or choose the Fit option. **Opposite corner -** Pick a point.
Horizontal	Draws three horizontal viewports. 
Vertical	Draws three vertical viewports. 
Above	Draws one viewport above two side-by-side viewports.

Below	Draws two side-by-side viewports above one viewport.
Left	Draws one viewport to the left of two stacked viewports. 
Right	Draws one viewport to the right of two stacked viewports. 
4	Draws four rectangular viewports; prompts you: **First corner of bounding rectangle or [Fit]** - Pick a point or choose the Fit option. Opposite corner - Pick a point. 
Object	Converts an object into a viewport border; prompts you: Select an object to clip viewport - specifies a closed object. The viewport is traced over the object; the original object remains in the drawing. This option is useful for ensuring a viewport border precisely matches an outline.

	
Polygonal	Draws non-rectangular viewports made of lines and arcs; prompts you: Specify start point - specifies the starting point for the polygon. Set next point or [draw Arcs/Distance/Follow] - specify a point, or enter an option. Set next point or [draw Arcs/Distance/Follow/Undo] - specify a point, or enter an option. Set next point or [draw Arcs/Close/Distance/Follow/Undo] - specify a point, enter an option, or press Enter to close the polyline. • Draw Arcs: Enters arc drawing mode; prompts you: Set end of arc or [draw Lines/Angle/CEnter/Direction/Radius/Second point] - enter an option (See Arc command). Use the Draw Lines option to exit arc drawing mode. • Close: Closes the polygon and exits the command. • Distance: Draws the next line segment at a specified distance and angle; prompts you: Distance for segment - type a value. Angle of segment - type a value. • Follow: Draws the next line segment at the same angle; prompts you: Set next point or [draw Arcs/Distance/Follow/Undo] - pick a point or enter an option. • Undo: Undoes (undraws) the last segment.

Related Commands

MvSetup - creates multiple paper space viewports; in model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.
MSpace - switches to model space.
PSpace - switches to paper space.
VpClip - clips viewports.
VPorts - creates viewports in model space.
RedrawAll - redraws all viewports.
RegenAll - regenerates the drawing in all viewports.
Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

MvSetup

Creates multiple paper space viewports; aligns, rotates and scales paper space viewports.
In model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.

Accessing the Command

command bar: *mvsetup*

: ***mvsetup***

Prompts you in the command bar:

When in model space:

Enable paper space? [No/Yes] <Yes>: (Press Enter to switch to the most recently used layout, choose No to continue in model space.)

Enter units type [Scientific/Decimal/Engineering/Architectural/Metric]: (Choose a unit type.)
Example of Metric scales: 2.0 is two times larger

Enter scale: (Type the scale factor)

Enter paper width: (Type the paper width in drawing units.)

Enter paper height: (Type the paper height in drawing units.)

A rectangle, representing the paper sheet with respect to the specified scale, is drawn. The lower left corner of the rectangle is placed at the origin of the current coordinate system.

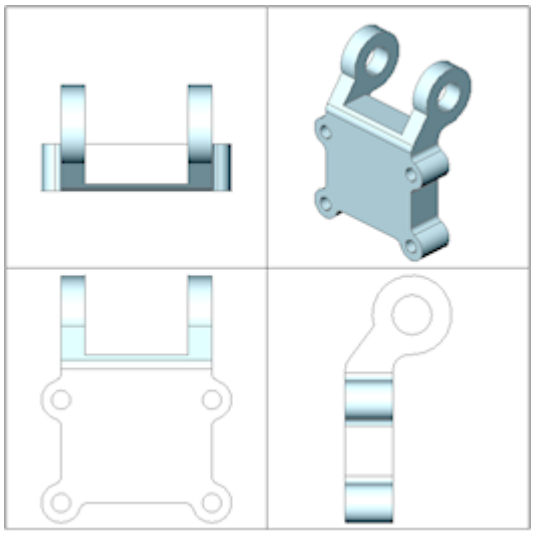
When in paper space:

Enter **an option [**Align**/**Create**/**Scale viewports**/**Undo**]:** (Enter an option.)

Command Options

Option	Description
Align	Pans the display in a viewport in order to align it with a base point in a reference viewport. Depending on the relative position of the two viewports, the display of the panned viewport might be shifted outside of the viewport boundary.
Angled	Specify basepoint: Pick a point in the reference viewport. Specify point in viewport to be panned: Pick a point, which will be used as the anchor point. Specify offset distance from base: Specify the desired distance between the basepoint and the anchor point. Either type the distance or pick two points. Specify angle from base: Specify the desired angle of the line between the basepoint and the anchor point. Either type the angle or pick two points. The display in the second viewport is panned over the specified distance and angle.
Horizontal	Specify basepoint: Pick a point in the reference viewport. Specify point in viewport to be panned: Pick a point, which will be used as the anchor point. The display in the second viewport is panned vertically to align the basepoint an the anchor point horizontally.

Vertical	Specify basepoint: Pick a point in the reference viewport. Specify point in viewport to be panned: Pick a point, which will be used as the anchor point. The display in the second viewport is panned horizontally to align the basepoint an the anchor point vertically .
Rotate view	Specify basepoint in the viewport to be rotated: Pick a point to define the basepoint of the rotation. Specify rotation angle: Either type the rotation angle or specify a second point. The display in the viewport is rotated over the specified angle. Note: If the VPROTATEASSOC system variable is ON, the display in a viewport is rotated if you rotate the viewport.
Undo	Undoes the previous alignment action and returns to the alignment options prompt.
Create	Creates viewports in the current layout.
Delete entities	Select entities: Select the entities you want to delete.
Create Viewports	**Creates new viewports, you are prompted to choose between 4 options:** Enter choice for layout options <0-3> [None/Single/Std. Engineering/Array of viewports] <0>: • 0 = None • 1 = Single viewport Specify first corner of bounding area for viewport(s): Pick a point. Other corner of rectangle: Pick a point. • 2 = 4 standard engineering viewports: Top, Front, Right and SE Isometric. Specify first corner of bounding area for viewport(s): Pick a point. Other corner of rectangle: Pick a point. Specify distance between viewports in X direction <0.000>: Type the desired horizontal spacing distance between the viewports. Specify distance between viewports in Y direction <0.000>: Type the desired vertical spacing distance between the viewports.

	 • 3 = Array of viewports Specify first corner of bounding area for viewport(s): Pick a point. Other corner of rectangle: Pick a point. Enter number of viewports in X direction <1>: Type the number of viewports. Enter number of viewports in Y direction <1>: Type the number of viewports. Specify distance between viewports in X direction <0.000>: Type the desired horizontal spacing distance between the viewports. Specify distance between viewports in Y direction <0.000>: Type the desired vertical spacing distance between the viewports.
Undo	Undoes the creation of viewports.
Scale Viewports	Select entities: Select the viewports you want to scale. **Entities in set: 20** Select more viewports or press Enter to stop selecting viewports. Set the scaling mode of viewports: Interactively/<Uniform>: Press Enter to scale all viewports uniformly or choose Interactively to set the scale for each viewport separately. Enter the number of paper space units <1.0>: Type a number. Enter the number of model space units <1.0>: Type a number.
Undo	Undoes all operations executed by the current MVSETUP command session.

Procedures

Creating an array of layout viewports
Setting the scale of a selection of viewports
To rotate the display in a viewport
To align viewports

Related Commands

AlignSpace - adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.
MView - creates one or more viewports in layout tabs.
MSpace - switches to model space.
PSpace - switches to paper space.
VpClip - clips viewports.
VPorts - creates viewports in model space.
RedrawAll - redraws all viewports.
RegenAll - regenerates the drawing in all viewports.
Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Mechanical

-BmParameters

Allows to list and edit the parameters of inserted components and associative array.

Accessing the Command

command bar: *-bmparameters*

: ***-bmparameters***
Prompts you in the command bar:

Select component instances or arrays: (Select a component or an associative array.)

Entities in set 1

Select component instances or arrays: (Select more components or associative arrays, or press Enter.)

Enter an option to change parameter expression [Edit/Delete/Link to parameter/?] <?>: (Press Enter to list the parameters of the selected components in the Prompt History window.)

Command Options

Option	Description
Edit	Allows to edit the value of a parameter. Prompts you: Enter parameter name: Type the name of a parameter. Enter expression: Type a value in the command bar.
Delete	Allows to reset a previously edited parameter to its default value. Prompts you: Enter parameter name: Type the name of a parameter.
Link to parameter	Assigning parameters as an expression for a parameter of a dimensional constraint (both 2D and 3D) associated with an entity which has been put into an associative array. Prompts you: Enter parameter name ('*' to link all) <*>: Type the name of a parameter.
?	Lists the parameters of the selected components in the Prompt History window. Press the F2 function key to display the Prompt History window.

Related Commands

ArrayPath - associatively distributes entity copies evenly along a path into multiple rows, and levels.
ArrayPolar - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
ArrayRect - associatively distributes entity copies into any number of rows, columns, and levels.
-DesignTableEdit - allows to edit designtables.
DesignTable - creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser and parametric component properties.
-Parameters - manages dimensional constraint parameters in the current drawing.

BmBalloon

Creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
Each balloon contains an index of the component in the associated BOM-table created by BmBom command.

Accessing the Command

command bar: *bmballoon*
menu bar: *Assembly | balloon*
toolbar: *Assembly |*
ribbon: *Assembly | Inquire | Balloon* and *Annotate | Assembly | Balloon*

: bmballoon

Prompts you in the command bar:

Select a component insert [select other Table/choose balloon Frame]: (Click a component insert.)

Pick point to place balloon**:** (Define the location of the balloon.)

Select a component insert [select other Table/choose balloon Frame]: *(Click the next component or press Enter to stop.)*

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.

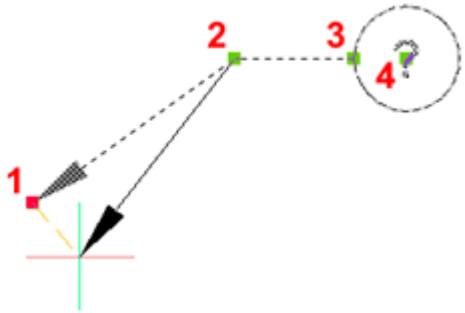
Command Options

Option	Description
Select a component insert	Selects the component and sets the location of the arrowhead.
Pick point to place balloon	Locates the balloon; prompts you: Pick point to place balloon - pick a point. Hold down the Shift-key to align the balloon with the previous one.
Select other table	Allows to select a different BOM-table in case multiple BOM-tables exist in the layout. The most recently insert BOM-table is selected by default; prompts you: Select a BOM table <Back> - Select the BOM-table or press Enter to proceed with the currentlye selected BOM-table.
Choose balloon frame	Allows to define the shape of the balloon; prompts you: Choose balloon frame [Circular/Rectangular/Triangular/Hexagonal/CUrrent] - choose an option.

Grips Editing

1. Select the balloon in the drawing.
   For a single segment balloon entity, four grips display: Endpoint (1), Vertex (2), Landing (3) and Text (4).
2. Drag the grips to perform the following actions:

- Move the arrowhead (1)
  If move the arrowhead to a different component, the balloon automatically updates the position number according to the associated BOM-table. If the arrowhead hits a blank space, the a question mark displays.
- Move the landing and the balloon (2)
- Edit the landing length(3)
- Move the balloon (4)



Related Commands

BimTag - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.
BmBom - inserts a Bill of Materials (BOM) table in the current drawing.
Ddedit - edits single-line text, multi-line text, attribute definitions, and attribute text
DimReassociate - reassociates or associates dimensions to entities or points on entities.
MLeader - creates multileader entities using the current multileader style.
MLeaderEdit - adds leader lines to and removes leader lines from a multileader entity.
MLeaderStyle - creates and manages multileader styles through the Drawing Explorer.

BmBom

Inserts a Bill of Materials (BOM) table in the current drawing.

Accessing the Command

command bar: *bmbom*

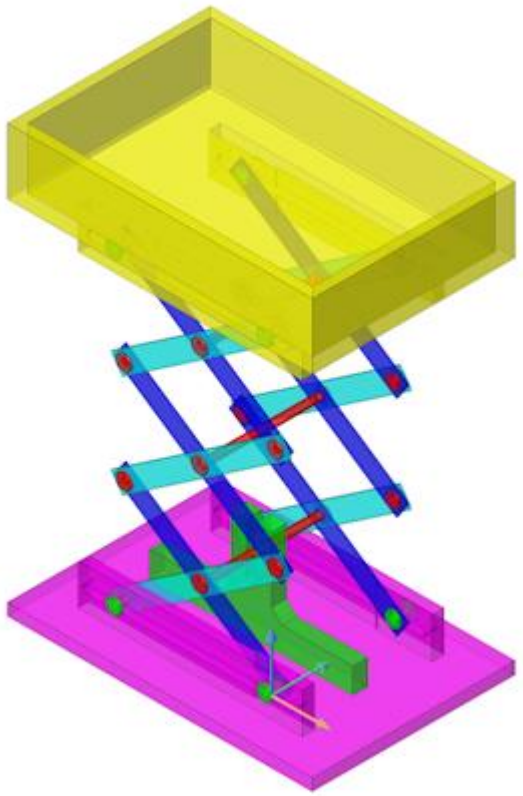
menu bar: *Assembly | Bill Of Materials*

ribbon: *Assembly | Inquire | Bill Of Materials*

toolbar: *Assembly |*

: ***bmbom***
Prompts you in the command bar:

Insertion point [Name/Top level/Bottom level/Configure]: *(Specify the insertion point or enter an option.)*



Command Options

Option	Description
Name	Prompts you to type a name in the command bar: Table name <Default Name>: Type the name, then press Enter. Prompts you: Insertion point [Name/Top level/Bottom level]: Specify the insertion point or enter an option.
Top level	Creates a BOM of the top level components only.

<table>
<tr><td></td><td>
<table>
<tr><th colspan="3">Bill of Materials lift</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>axis_b</td><td>4</td></tr>
<tr><td>2</td><td>axis_m</td><td>3</td></tr>
<tr><td>3</td><td>axis_s</td><td>4</td></tr>
<tr><td>4</td><td>bar_D</td><td>6</td></tr>
<tr><td>5</td><td>bar_U</td><td>6</td></tr>
<tr><td>6</td><td>base_assy</td><td>1</td></tr>
<tr><td>7</td><td>connector</td><td>8</td></tr>
<tr><td>8</td><td>platform_U</td><td>1</td></tr>
</table>
</td></tr>
<tr><td>Bottom level</td><td>
Creates a BOM of the subcomponents.
<table>
<tr><th colspan="3">Bill of Materials lift</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>axis_b</td><td>4</td></tr>
<tr><td>2</td><td>axis_m</td><td>3</td></tr>
<tr><td>3</td><td>axis_s</td><td>4</td></tr>
<tr><td>4</td><td>bar_D</td><td>6</td></tr>
<tr><td>5</td><td>bar_U</td><td>6</td></tr>
<tr><td>6</td><td>base</td><td>1</td></tr>
<tr><td>7</td><td>bracket</td><td>1</td></tr>
<tr><td>8</td><td>connector</td><td>8</td></tr>
<tr><td>9</td><td>platform_U</td><td>1</td></tr>
</table>
Compared to the Top Level BOM, the base_assy component is broken up into the base and bracket components.
</td></tr>
<tr><td>Configure</td><td>
Allows to add additional columns to your BOM table: Description, Density, Volume, Mass, Material, Thickness and Parameters (for parametric components).
Prompts you:
Toggle columns [Description/dEnsity/Volume/Mass/parameterS/mAterials/Thickness/Back] <Back>:
Choose an option:

Description: adds the Description column.
Density: adds the Density column.
Volume: adds the Volume column.

</td></tr>
</table>

- **Mass:** adds the Mass column.
- **Parameters:** If several inserts of the same parametric component exist in the assembly they will be grouped in different BOM rows depending on the values of their parameters.
- **Materials:** adds the Material column.
- **Thickness:** adds the Thickness column. If a component is a sheet metal part, this column will show the thickness of this part. Otherwise, it will be empty.
- **Back:** Returns to the previous prompt.

Bill of Materials lift

No.	Component	Quantity	Density	Volume	Mass	base_height	base_lenght	base_width	bracket_heigh
1	axis_b	4	1.0000	1459.7712	1459.7712				
2	axis_m	3	1.0000	6787.9345	6787.9345				
3	axis_s	4	1.0000	1359.2494	1359.2494				
4	bar_D	6	1.0000	7698.4189	7698.4189				
5	bar_U	6	1.0000	7698.4189	7698.4189				
6	base	1	1.0000	870208.5882	870208.5882	52.0000	300.0000	200.0000	
7	bracket	1	7.8560	115287.0878	905695.3620				100.00
8	connector	8	1.0000	957.1339	957.1339				
9	platform_U	1	1.0000	1029180.1368	1029180.1368				

BOM with Densitiy, Volume, Mass and Parameters

Related Commands

Table - creates a table entity in the drawing through a dialog box.
-Table - creates a table entity in the drawing through the command bar.
Tabledit - Edits texts in a table cell.

BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
BmExplode - creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location.
BmMassProp - computes mass properties for the current model using densities assigned its the components.
BmUpdate - reloads all referenced components from external files and updates BOM tables.

BmBrowser

Obsolete, use MechanicalBrowserOpen instead.

BmDependencies

Lists all files, containing component definitions inserted in the assembly, in the command window.

Accessing the Command

command bar: *bmdependencies*

: ***bmdependencies***
Reports in the command bar:

```
Found 10 dependencies for this assembly:
C:\MCAD\Lift\axis_b.dwg      - OK
C:\MCAD\Lift\axis_m.dwg      - OK
C:\MCAD\Lift\axis_s.dwg      - OK
C:\MCAD\Lift\bar_D.dwg       - OK
C:\MCAD\Lift\bar_U.dwg       - OK
C:\MCAD\Lift\base.dwg        - OK
C:\MCAD\Lift\base_assy.dwg   - NOT FOUND
C:\MCAD\Lift\bracket.dwg     - OK
C:\MCAD\Lift\connector.dwg   - OK
C:\MCAD\Lift\platform_U.dwg  - NOT FOUND
```

Command Options

No options.

Related Commands

MechanicalBrowserOpen - toggles the visibility of the *Mechanical Browser* window.
BmRecover - recovers a broken mechanical structure.
BmUpdate - reloads all referenced components from external files and updates BOM tables.

BmDissolve

Dissolves a mechanical component inserted in the current drawing.

Accessing the Command

command bar: *bmdissolve*
menu bar: *Assembly | Dissolve Component*
ribbon: *Assembly | Modify | Dissolve Component*
toolbar: *Assembly |*

: ***bmdissolve***
Prompts you in the command bar:

Select entities: (select an entity)

Entities/subentities in set: 1

Select entities: (select more entities or right click to stop selecting entities.)

Only components at the first level, directly branching from the root component, in the mechanical hierarchy can be dissolved. To dissolve a component at a lower level, its parent component(s) must be dissolved first.

All subcomponents of a dissolved component are moved up one level in the mechanical hierarchy.

Dissolving a component insert does not affect the source drawing of the component.

The selection set must contain component inserts only. If not, a message displays in the command bar: "No components selected."

Command Options

No options.

Related Commands

BmExternalize - converts local components to external components.
BmForm - creates a new mechanical component and inserts it into the current drawing.
BmInsert - inserts an existing mechanical component into the current drawing.
BmLocalize - converts external components to local components.
BmUnmech - converts the current mechanical component into a plain drawing.

BmExplode

Creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location.

It supports two levels of the exploded representation: top and bottom, as well as three possible algorithms: Table by Level, Table by Type and Manual. 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..

Accessing the Command

command bar: *bmexplode*
menu bar: *Assembly | Explode*
toolbar: *Assembly |*
ribbon: *Assembly | Tools | Explode*

: ***bmexplode***
Prompts you in the command bar:

Select explosion algorithm [Table by Level/Table by Types/Manual/Settings]: (Choose the algorithm or set the necessary options)

Select position of the exploded representation: (Choose the position to insert the exploded representation.)

Command Options

Option	Description
Table by Level	Selects the Table by Level. The Table by Level algorithm groups parts by their level in the assembly tree and puts the parts of the same level in the same row.
Table by Types	Selects the Table by Types. The Table by Types algorithm groups same parts on the same row and different parts on different rows.
Manual	Selects the manual mode, which creates the exact copy of the assembly ready for further editing in order to create a customized exploded view.
Settings	Allows to access the command settings; prompts you: Select explode level or set block name [Top/Bottom/set Name/BAck - choose an option • Top: If selected, the assembly will be exploded using only top level components.. • Bottom: If selected, the assembly will be exploded down to bottom level components. • Name: Allows to set the name for the exploded representation; prompts you: Name for exploded block: Type the name of the exploded representation - type a name. If a such name is already assigned to another exploded representation, a postfix to make it unique will be added, starting from '_2.' • BAck: Returns to the previous prompt.

Procedures

Levels of the exploded representation
Selecting an algorithms
To create an exploded representation
To generate drawing views of the exploded representation
To update an exploded representation

Related Commands

BmBalloon - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
BmBom - inserts a Bill of Materials (BOM) table in the current drawing.

BmExternalize

Converts local components to external components.

Accessing the Command

command bar: *bmexternalize*
menu bar: *Assembly | Switch Component to External*

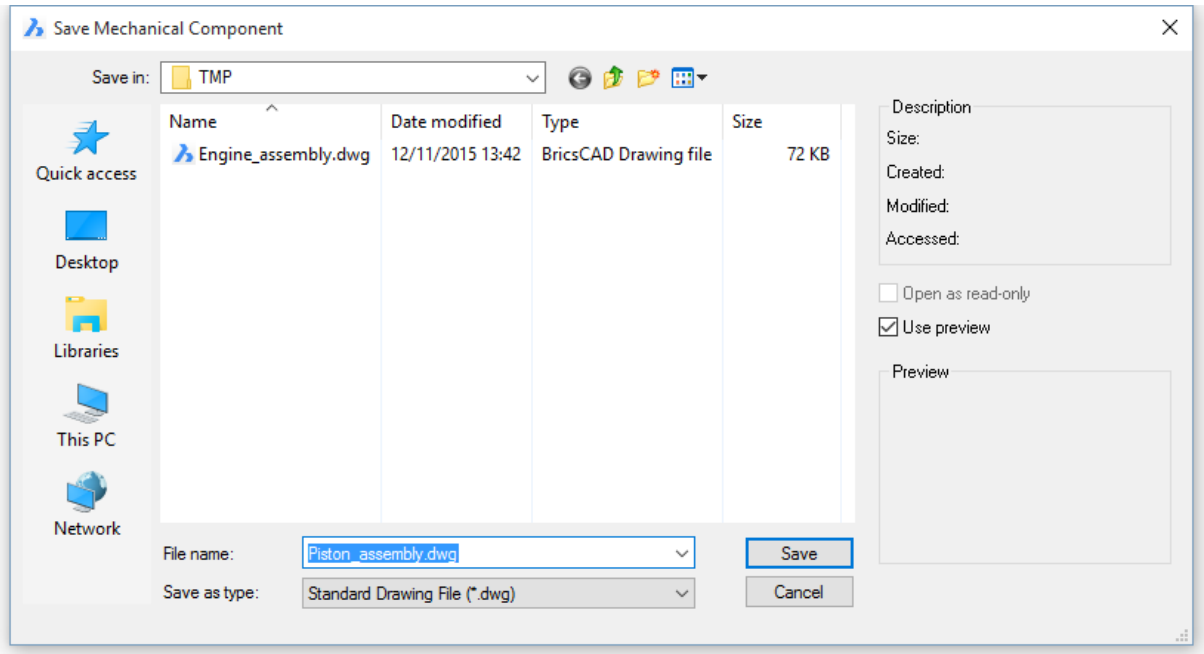
ribbon: *Assembly | Modify | External*
toolbar: *Assembly |*

: ***bmexternalize***
Prompts you in the command bar:
Select component instances [ASk for file names/Entire model] <Entire model>: *(Select a local component.)*

Entities in set: 1

Select component instances [ASk for file names/Entire model] <Entire model>: *(Select a local component or press Enter.)*

A file dialog displays:



Do one of the following:

- Press the Save button to accept the default name, which is the name of the selected local component.
- Type a name in the File Name field, then click the Save button.

This procedure is repeated for each of the selected local components.

Command Options

Option	Description
Entire Model	For each of the local components in the assembly a file dialog displays. Do one of the following: • Press the Save button to accept the default name, which is the name of the local component. • Type a name in the File Name field, then click the Save button.
Ask for file names	You can select to ask for file names of exernalized components or generate them automatically. Prompts you: Ask for file names of external components to be saved? [Yes/No] <Yes>: • Yes: You are prompted to enter file names. • No: File names are generated automatically. Please note that some symbols used in component names cannot be used in file names. Such symbols will be automatically replaced with an underscore (_).

Related Commands

BmDissolve - dissolves a mechanical component inserted in the current drawing.
BmLocalize - converts external components to local components.
BmReplace - replaces a component insert.

BmForm

Creates a new mechanical component and inserts it into the current drawing.

If necessary, run the BmMech command to initialize the mechanical structure in the current drawing.

Accessing the Command

command bar: *bmform*
menu bar: *Assembly | Form Component*
ribbon: *Assembly | Modify | Form Component*
toolbar: *Assembly |*

: ***bmform***
Prompts you in the command bar:

Select entities: (Select a 3D solid.)

Entities in set: 1
If a history based solid is selected, prompts you:
Selection set contains 3D solid/solids with recorded history. Confirm selection and purge history? <Yes>/No: (Press Enter to accept the default option to include the X-Solid in a mechanical component.)

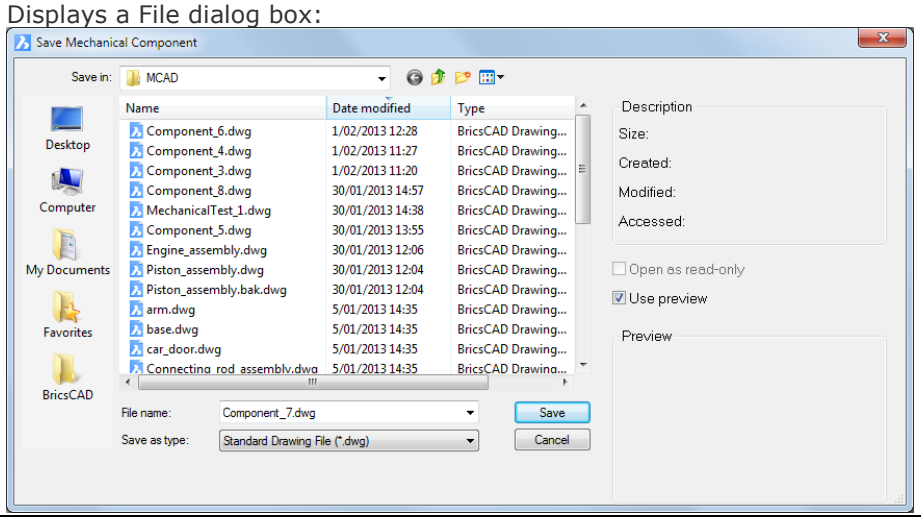
Select entities: (Select a 3D solid or press Enter to stop selecting 3D solids.)

Enter base point for new local component [**create External component**] <0, 0, 0>: (Specify a point.)

Name for new mechanical component <Component_1>: (Specify a name, or press Enter to accept the default name.)

A new internal block, containing the selected solid(s), is created and inserted as an internal reference in the current drawing. The selected solid(s) are removed from the current drawing.

Command Options

Option	Description
Create External component	Creates a new drawing, containing the selected solid(s). The selected solid(s) are removed from the current drawing. The new drawing is inserted as an external reference in the current drawing. Displays a File dialog box:  	

	Select a folder, type a name in the File Name field, then press the Save button

Related Commands

BmDissolve - dissolves a mechanical component inserted in the current drawing.
BmExternalize - switches internal components to external components.
BmInsert - inserts an existing mechanical component into the current drawing.
BmLocalize - switches external components to internal components.
mXConvert - converts X-Hardware solids in the current drawing to mechanical components.

-BmHardware

Inserts a standard hardware part as a mechanical component in the current drawing. Prompts you in the command bar.

If necessary, run the BmMech command to initialize the mechanical structure in the current drawing.

Accessing the Command

command bar: *-bmhardware*

: ***-bmhardware***

Prompts you in the command bar:

Input a category (? to list available categories) <ALL>: (Type a category name.)

Input a type (? to list available types) <ACORN-RVT>: (Specify a type.)

Input a size (? to list available sizes) <.500>: (Specify the size.)

Input a length (? to list available lengths) <1.0>: (Specify the length.)

Insertion point [**Rotate**/**Base point**/**Name**/**mUltiple**] <0,0,0>: (Specify a point in the drawing.)

Command Options

Option	Description
Rotate	Allows to specify a rotation angle. Prompts you: Rotation angle [Move/Base point/Name/mUltiple]: Type a value or choose an option.
Base point	Allows to specify a new base point for the insert. Prompts you: New base point <0,0,0>: Specify a point.
Name	Allows to rename the insert. Prompts you: Component insert name <current_name>: Type a name.
Multiple	Allows to insert multiple copies. Prompts you for each copy: Insertion point [Rotate/Base point/Name] <0,0,0>: Specify a point. Press Enter to stop inserting more copies.

Related Commands

StandardPartsPanelOpen - opens the Standard Parts panel, allowing to insert a standard hardware part as a mechanical component in the current drawing.

BmHardware

Obsolete, use StandardPartsPanelopen instead.

BmHide

Hides a mechanical component inserted in the current drawing.
The command affects the visibility of the selected inserts only. Hidden inserts will still be taken into account by commands such as BmBom, BmMassProp etc.

Accessing the Command

command bar: *bmhide*

menu bar: *Assembly Visualization | Hide*

ribbon: *Assembly I Modify | Hide*

toolbar: *Assembly Visualization* |

: ***bmhide***
Prompts you in the command bar:
Select insert or enter an option [Component name/Insert name]<Insert name>: *(Select an insert in the drawing or choose an option.)*

Command Options

Option	Description
Component name	Prompts you: Component name, all its inserts will be hidden: Type the component name of the component insert(s) you want to hide. All inserts of the specified component will be hidden.
Insert name	Prompts you: Component insert name, use / to separate names of nested inserts: Type the insert name of the component insert you want to hide. If you want to hide a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/)

Related Commands

BmBrowser - toggles the visibility of the *Mechanical Browser* window.
BmShow - shows a previously hidden mechanical component or subcomponent in the current drawing.
BmVStyle - applies a visual style to a mechanical component.

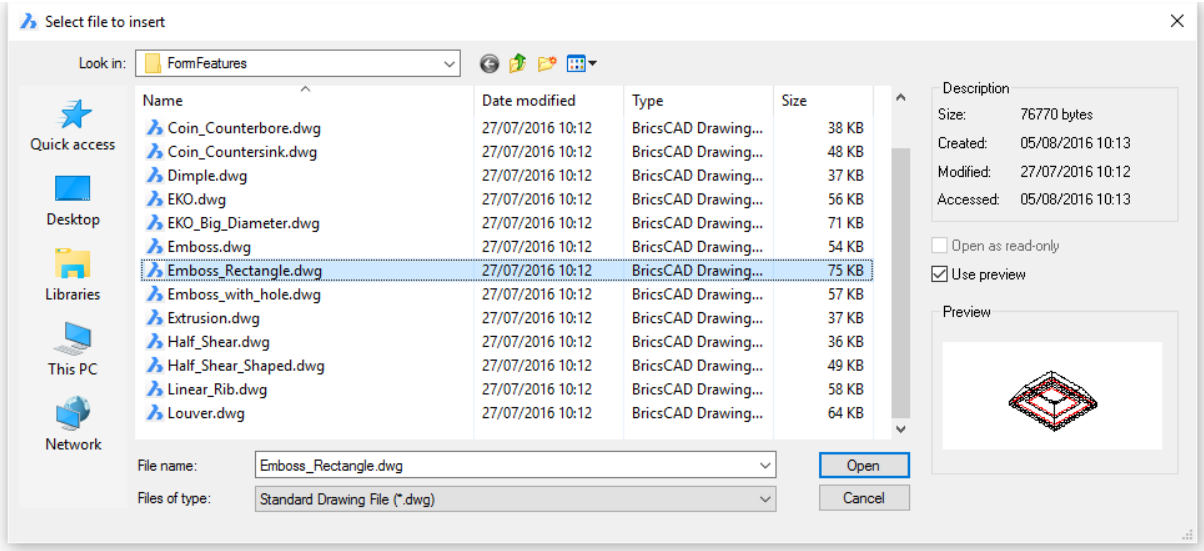
BmInsert

Inserts an existing mechanical component or sheet metal form feature into the current drawing. The source drawing of the component is not affected. A virtual component is created, representing the drawing in the assembly.

Accessing the Command

command bar: *bminsert*
menu bar: *Assembly* | *Insert Component* and *Sheet Metal* | *Insert Form Feature*
ribbon: *Assembly* | *Modify* | *Insert Component* and *Sheet Metal* | *Modify* |
toolbar: *Assembly* | and *Sheet Metal* |

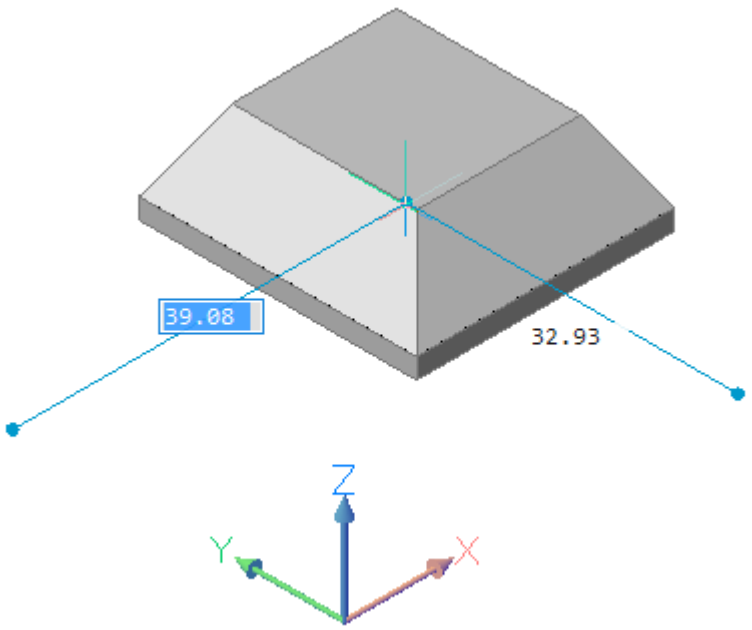
: ***bminsert***
Displays a dialog box:



Select a mechanical component drawing, then double click or press the *Open* button.

The component is attached to the cursor with its origin point.

Dynamic dimensions display: X- and Y coordinates from the origin of the current coordinate system.



Prompts you in the command bar:

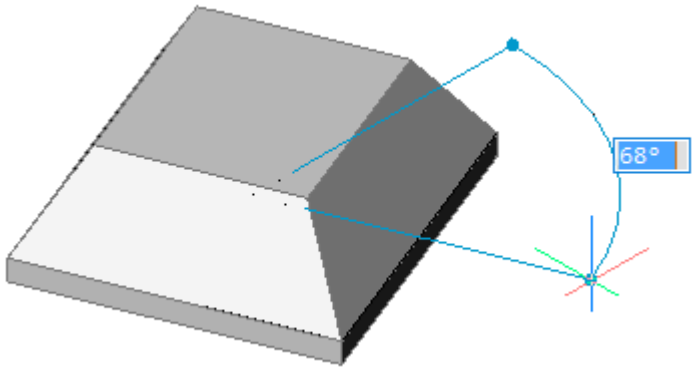
Select insertion point or [Edit inserted entity/Rotate/Base point/Name/Insertion Type/Flip/mUltiple] <0,0,0>: (Specify the insertion point or choose an option.)

Command Options

Option	Description

Edit inserted entity	Allows to edit the parameters of the selected insert.
	Prompts you:
	Enter parameter name or press ENTER to finish:
	Press F2 to open the Prompt History window.
	`----------`
	`Parameter: _SM_THICKNESS Expression: Value: 2.00 mm`
	`Parameter: A Expression: Value: 30.00`
	`Parameter: L Expression: Value: 16.00 mm`
	`Parameter: W Expression: Value: 14.00 mm`
	`Parameter: H Expression: Value: 8.00 mm`
	`----------`
	Type the name of a parameter.
	Prompts you:
	Enter expression < 30.00 >: Type a value, then press Enter.
	Prompts you:
	Enter parameter name or press ENTER to finish: Type the name of a parameter or press Enter.
	The insert is updated.

Rotate	Allows to define the rotation angle of the component. Prompts you: Specify rotation angle of the insert: Enter the rotation angle.

	
Base point	Allows to specify a different base point to insert the component. Prompts you: New base point <0,0,0>: Specify a new base point. The new base point is attached to the cursor.
Name	Allows to define the insert name of the component. Prompts you: Component insert name <DefaultName>: Type the insert name for the component.
Insertion type	Allows to specify the insertion type: Local or External Prompts you: Specify component type [Local/External] <External>: Choose an option.
Flip	Inserts the selected insert in the opposite face of the solid.
Multiple	Allows to insert multiple inserts of the same component or create an array. Prompts you: Select insertion point or [Edit inserted entity/Rotate/Base point/Name/Insertion Type/Flip/Array]: Optionally, choose an option, then do one of the following: • Select multiple insertion points. Press Enter to stop inserting more copies. • Choose Array: Prompts you: Specify base point for array: Specify a point. Enter columns spacing or set [Direction]: Specify a distance or choose Direction. Enter the distance between rows or set [Single row/Rectangular/Direction]: Specify a distance or choose an option. Select an end point for creating array: Move the cursor to dynamically define the extents of the array, then click a point. Accept resulting array or edit [Base point/Columns/Rows] <Accept>: Press Enter to accept the array or choose an option.

Related Commands

BmDissolve - dissolves a mechanical component inserted in the current drawing.
BmForm - creates a new mechanical component and inserts it into the current drawing.
BmHardware - inserts a standard hardware solid as a mechanical component in the current drawing.
BmMech - converts the current drawing into a mechanical main component.
BmUnlink - breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.
DmCopyFaces - copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.
SmReplace - replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.
SmRibCreate - creates associative Rib form features on sheet metal parts from 2D profiles.

BmLocalize

Converts external components to local components.

Accessing the Command

command bar: *bmlocalize*
menu bar: *Assembly | Switch Component to Local*
ribbon: *Assembly | Modify | Local*
toolbar: *Assembly |*

: ***bmlocalize***
Prompts you in the command bar:

Select component instances/<Entire model>: (Select an external component.)

Entities in set: 1

Select component instances/<Entire model>: (Select an external component or press Enter to convert the selected component(s) to local components.)

If multiple inserts of the same external component exists in the model, all inserts are converted to local components.

Command Options

Option	Description
Entire model	Converts all external components in the model to local components.

Related Commands

BmDissolve - dissolves a mechanical component inserted in the current drawing.
BmExternalize - converts local components to external components.
BmReplace - replaces a component insert.

BmMassProp

Computes mass properties for the current model using densities of materials assigned to its the components.
The density values used for the mass property calculations are taken from materials defined by the *Material* property of the components and subcomponents, which can be either inherited from the main component of the assembly or set explicitly. The density of the main component is defined by the materials set in the *Material* field in the Mechanical Browser dialog. If a material does not have an assigned density, or its density is less or equal to zero, an error message displays, and the mass properties will not be computed. All materials involved in the assembly must have non-zero non-negative densities.

Accessing the Command

command bar: *bmmassprop*
menu bar: *Assembly | Mass Properties*
ribbon: *Assembly | Inquire | Mass Properties*
toolbar: *Assembly |*

: ***bmmassprop***
Prompts you in the command bar:

Select 3D solids and subcomponents or [Entire model] <Entire model>: **(Select an item.)**

Entities in set: 1

Select 3D solids and subcomponents or [Entire model] <Entire model>: **(Select more items or right click.)**
Reports in the Prompt History window:

---------- Component ----------

Mass: 14.1628 kg

Volume: 3463.1024 cm³

Centroid: X= 24.5717 mm Y= 0.0467 mm Z= -58.4273 mm

Moments of inertia: X= 0.1695 kg·m² Y= 0.1859 kg·m² Z= 0.2332 kg·m²

Products of inertia:

XY: 0.0001 kg·m²

YZ: -0.0000 kg·m²

ZX: -0.0240 kg·m²

Radii of gyration: X= 109.4107 mm Y= 114.5675 mm Z= 128.3054 mm

Principal moments and X-Y-Z directions about centroid:

I: 0.1211 kg·m² along X= 0.9994 Y= 0.0053 Z= -0.0356

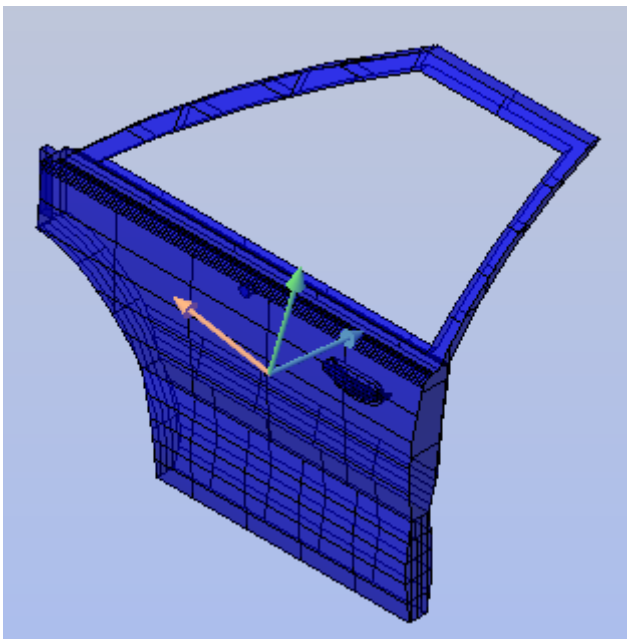
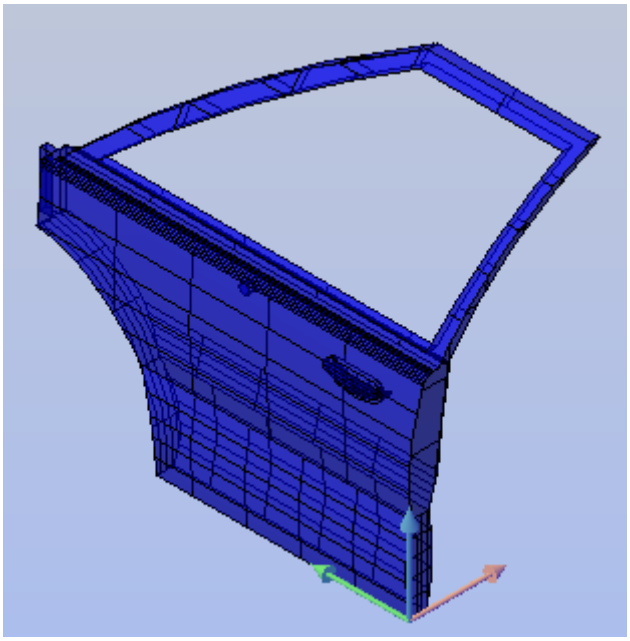
J: 0.1290 kg·m² along X= -0.0053 Y= 1.0000 Z= 0.0003

K: 0.2247 kg·m² along X= 0.0356 Y= -0.0001 Z= 0.9994

Keep UCS aligned with principal moment axes? Yes/<No>: (Press Enter or right click to keep the current UCS)

NOTE BmMassProp will format the values of mass, volume and linear nature with their corresponding units only if the PROPUNITS system variable has the corresponding bits set to 1, or the corresponding flag in the Settings dialog for the Property units field is checked. See To set the Mass Units.

Command Options

Option	Description
Keep UCS aligned with principal moment axes? Yes / No	 **Yes**: Aligns the UCS with the principal moment axes.  **No**: Keeps the current UCS.

Related Commands

BmBom - inserts a Bill of Materials (BOM) table in the current drawing.
BmBrowser - toggles the visibility of the Mechanical Browser window.

BmMech

Converts the current drawing into a mechanical component.
If the drawing contains block references or external references, they can be optionally converted into inserts of local and external components correspondingly.
The command does nothing if the drawing is a mechanical component already.

Accessing the Command

command bar: *bmmech*
menu bar: *Assembly| Initialize Mechanical Structure*
ribbon: *Assembly | Create | Initialize Mechanical Structure*
toolbar: *Mechanical |*

: ***bmmech***
Prompts you in the command bar:

Name for new mechanical component <CurrentDrawingName>: (Press Enter to accept the current name or type a new name if you want to keep the original. If the drawing was not yet saved before, a default name, Component_1, is suggested.)

Convert blocks and external references to mechanical components? [No/Yes] <Yes>: (Press Enter or choose an option.)

Command Options

Option	Description
Yes	If the drawing contains block references or external references, blocks are converted to internal components, external references are converted to external components. Blocks are no longer avaialbe in the INSERT command. Use BMINSERT to insert them as internal components.
No	The mechanical structure is initialized, but the blocks and external references are not converted.

Related Commands

BmExternalize - converts local components to external components.
BmInsert - inserts an existing mechanical component into the current drawing.
BmLocalize - converts external components to local components.
BmNew - creates a new mechanical component as a new drawing file.
BmUnmech - converts the current mechanical component into a plain drawing.

BmNew

Creates a mechanical component as a new drawing file.

Accessing the Command

command bar: *bmnew*
menu bar: *Assembly | New Component*
ribbon: *Assembly | Create | New Component*
toolbar: *Assembly |*

: ***bmnew***
Prompts you in the command bar:
Name for new mechanical component <Component_x>: *(Type a name for the new component or press Enter to accept the default name.)*

Command Options

No options.

Related Commands

BmMech - converts the current drawing into a mechanical main component.
BmUnmech - converts the current mechanical component into a plain drawing.

BmOpen

Opens the source drawing of an external mechanical component.

Accessing the Command

command bar: *bmopen*
menu bar: *Assembly | Open Component File*
ribbon: *Assembly | Modify | Open*
toolbar: *Assembly |*

: ***bmopen***
Prompts you in the command bar:
Select insert: *(Select an external component in the model or the Mechanical Browser.)*

Command Options

No options.

Related Commands

BmOpenCopy - opens a copy of a component insert as a new drawing.
XOpen - opens an externally-referenced drawing (Xref) for editing in a separate window.

BmOpenCopy

Opens a copy of a component insert as a new drawing.

The command can be used to create a drawing of a local component. You can then use the BmReplace command to replace the original local component with the newly created drawing. If you open a copy of an insert of a parametric component the current values of the various parameters are applied to the copy.

Accessing the Command

command bar: *bmopencopy*
menu bar: *Assembly | Open Component Copy*
ribbon: *Assembly | Modify | Open Copy*
toolbar: *Assembly |*

: ***bmopencopy***
Prompts you in the command bar:

Select insert: (Select an external component in the model or the *Mechanical Browser*.)

A new drawing: <component_name>1 is created. Use the SAVE command to rename and save the drawing.

Command Options

No options.

Related Commands

BmOpen - opens the source drawing of an external mechanical component.

BmRecover

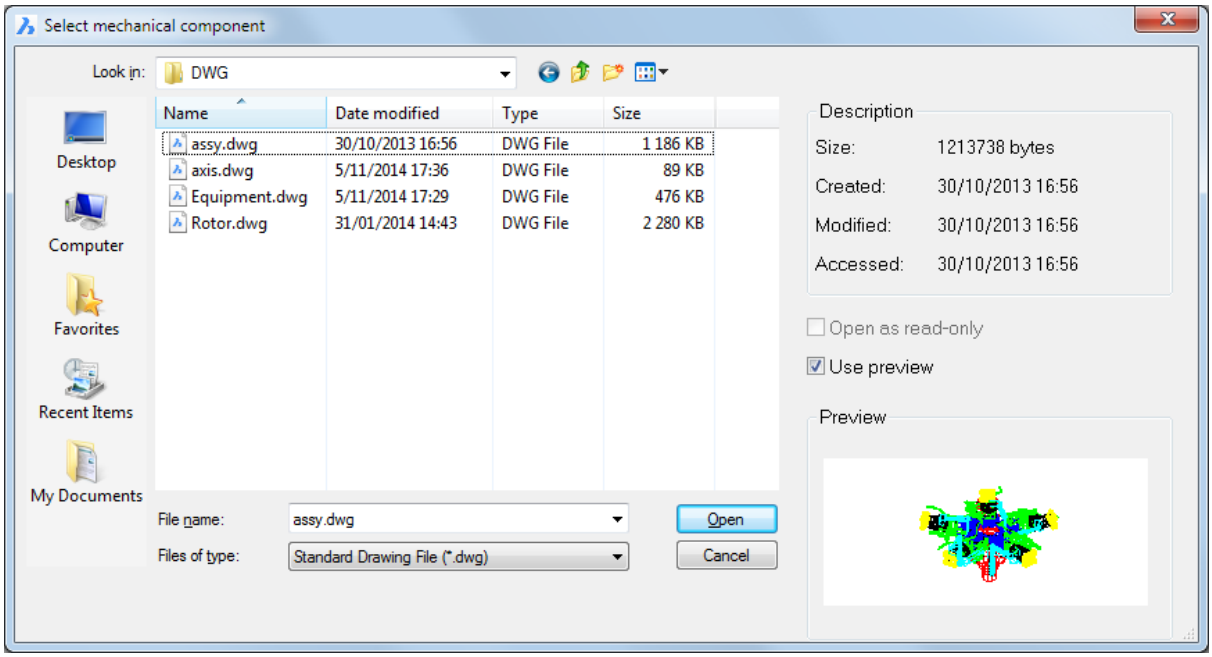
Recovers a broken mechanical structure.
The command is different from the basic Audit and Recover commands. It is recommended to use BmRecover to mend issues with a MCAD drawing. The command overwrites the original drawing.

Accessing the Command

command bar: *bmrecover*
menu bar: *Assembly | Recover*
ribbon: *Assembly | Tools | Recover*
toolbar: *Assembly |*

: ***bmrecover***

Displays a dialog box:



(Select a file, then double click or click the Open button.)

The command reports in the command window:

Fixing drawing.
...
Saving...
Done.

Command Options

No options

Related Commands

BmDependencies - lists all files, containing component definitions inserted in the assembly, in the command window.
BmUpdate - reloads all referenced components from external files and updates BOM tables.

BmReplace

Replaces a component insert.

Accessing the Command

command bar: *bmreplace*
menu bar: *Assembly | Replace Component*

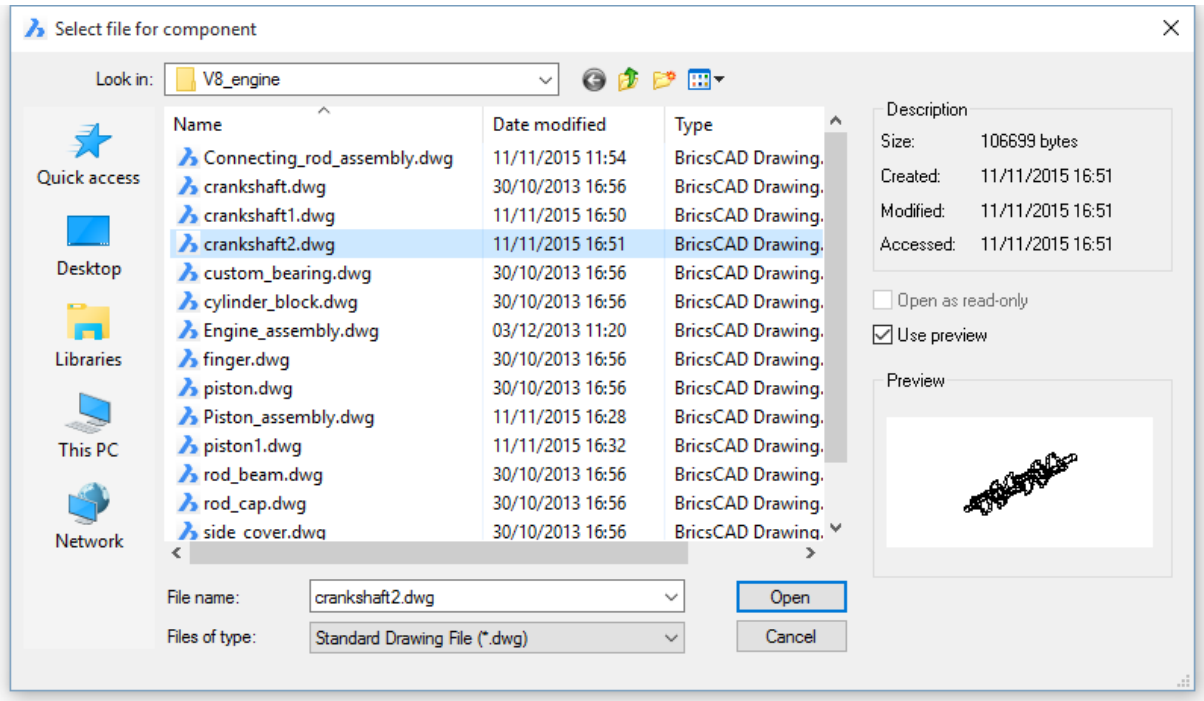
ribbon: *Assembly | Modify | Replace*
toolbar: *Assembly |*

: ***bmreplace***
Prompts you in the command bar:
Select component inserts [SIMilar inserts/component Type]: *(Select an insert in the drawing.)*

Entities in set: 1

Select component inserts [SIMilar inserts/component Type]: *(Select an insert in the drawing or press Enter.)*

A File dialog displays:



Select a file, then double click or click the Open button.

Command Options

Option	Description
Similar Inserts	Allows to replace similar inserts automatically. Prompts you: Replace file for all similar inserts [Yes/No] <Yes>: • Yes: Replaces all similar inserts of the selected mechanical component(s) in your assembly.

	• No: Replaces the selected component(s) only.
Component Type	Allows to choose the insertion type for replacement component. See External and local components in the User Guide. Prompts you: Component type [Local/External/Keep as is/component Default] <component Default>: • 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.

Related Commands

BmExternalize - converts local components to external components.
BmOpen - opens the source drawing of an external mechanical component.

BmShow

Shows previously hidden mechanical components inserted in the current drawing.

Accessing the Command

command bar: *bmshow*

menu bar: *Assembly Visualization | Show*

ribbon: *Assembly | Modify | Show*

toolbar: *Assembly Visualization* |

: ***bmshow***
Prompts you in the command bar:
Select insert or enter an option [All/Component name/Insert name]<All>: *(Select a component of which some inserts are currently hidden or choose an option.)*

Command Options

Option	Description
All	Shows all component inserts.
Component name	Prompts you: Component name, all its inserts will be shown: Type the component name of the component insert(s) you want to show. This affects all inserts of the specified component.
Insert name	Prompts you: Component insert name, use / to separate names of nested inserts: Type the insert name of the component insert you want to show. If you want to show a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/)

Related Commands

BmBrowser - toggles the visibility of the *Mechanical Browser* window.
BmHide - hides a mechanical component or subcomponent in the current drawing.
BmVStyle - applies a visual style to a mechanical component.

BmUnlink

Breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.

Accessing the Command

command bar: *bmunlink*

: ***bmunlink***
Prompts you in the command bar:
Select component instances: *(Select a component.)*
Entities in set: 1
Select component instances: *(Select another component or press Enter.)*

Command Options

No command options.

Related Commands

BimInsert - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.
BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.

BmUnmech

Converts the current mechanical component into a plain drawing.
The command applies to drawings that are a mechanical component only.

Accessing the Command

command bar: *bmunmech*
menu bar: *Assembly | Remove Structure*
ribbon: *Assembly | Tools | Remove Structure*
toolbar: *Assembly |*

: ***bmunmech***
Prompts you in the command bar:
Warning: this command removes all mechanical structure from the current document. Proceed?
No/<Yes>: *(Press Enter or choose an option)*

Command Options

Option	Description
Yes	Removes the mechanical structure from the drawing, thus turning it into a plain drawing. Inserts of local components are converted to block references (the component name becomes the block name); inserts of external components are converted to external references.
No	Aborts the command.

Related Commands

BmDissolve - dissolves a mechanical component inserted in the current drawing.
BmMech - converts the current drawing into a mechanical component.
BmNew - creates a mechanical component as a new drawing file.

BmUpdate

Reloads all referenced components from external files and updates BOM tables.

Use this command to

- Reload referenced components from external files (BMINSERT)
- Rebuild component based features, sheet metal form features and BIM windows/doors components (see about BC_SUBTRACT layer, Form Features, BIMINSERT)
- Update the Bill Of Material (BMBOM)
- Regenerate 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.

Accessing the Command

command bar: *bmupdate*
menu bar: *Assembly | Update* and *Update Selected*
ribbon: *Assembly | Tools | Update* and *Update Selected*
toolbar: *Assembly |* and

: ***bmupdate***
Prompts you in the command bar:
Select entities to update or [MOde/Entire model] <Entire model>: (Select a component insert or generated drawing view; or select one of the faces of the component based or sheet metal form feature; or press Enter to select the entire model.)
Entities in set: 1
Select entities to update or [MOde/Entire model] <Entire model>: (Select more objects.)
Updated xx definitions.

Command Options

Option	Description
Mode	Allows to change the way how the new placement face is defined. Each inserted component-based feature or BIM Window component has an associated placement face of the correspondent 3D solid the feature is attached to. This is the face which was used for feature placement in the BMINSERT command. The associated placement face of the feature can be lost in modelling or copying operations or it can be absent if the feature has been inserted in an empty area. The BMUPDATE command allows to re-associate the feature with its placement face in one of two modes. Prompts you: **Select placement face [Automatic/Manual] <Automatic>:** • **Automatic** - the placement face for the feature is detected automatically • **Manual** - prompts you to select the new placement face
Entire model	Updates the entire model.

Related Commands

BmBom - inserts a Bill of Materials (BOM) table in the current drawing.
BmBrowser - toggles the visibility of the *Mechanical Browser* window.
BmDependencies - lists all files, containing component definitions inserted in the assembly, in the command window.
BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.
BimInsert - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.

BmVStyle

Applies a visual style to a mechanical component insert.

Accessing the Command

command bar: *bmvstyle*
menu bar: *Assembly Visualization / Visual Style*
ribbon: *Assembly / Modify | Visual Style*
toolbar: *Assembly |*

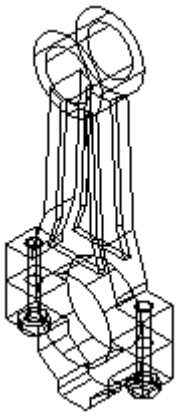
: ***bmvstyle***
Prompts you in the command bar:

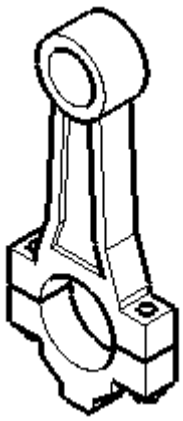
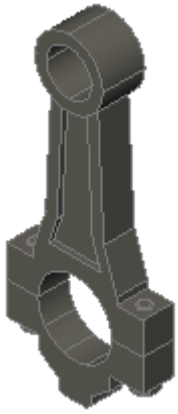
Select visual style [by Viewport/3d wireframe/Hidden/Realistic/Conceptual/Modeling/high Quality/by Name] <3d wireframe>: (Choose an option.)

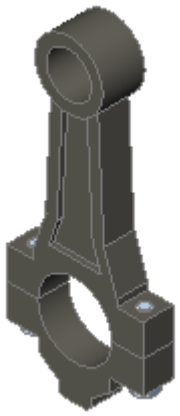
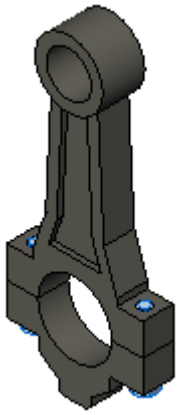
Select insert or enter an option [All/name of Insert]<All>: (Select an option or select a component insert - press Enter to select all components.)

Component insert name, use / to separate names of nested inserts: (Type the names of one or more component inserts separated by slashes (/). Component insert names are case sensitive.)

Command Options

Option	Description
By Viewport	Applies the current visual style of the viewport.
3D 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.

	
High Quality	Applies the *High Quality* visual style. 
By Name	Applies a named visual style.

Related Commands

Hide - removes hidden lines from 3D entities.
Render - generates a photorealistic rendering using materials and lights through a dialog box.
-Render - generates a photorealistic rendering using materials and lights through the command bar.
Shade - generates shaded images of 3D drawings.
Shademode - Sets the shading style for 3D drawings.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

BmBrowser - toggles the visibility of the *Mechanical Browser* window.
BmHide - hides a mechanical component or subcomponent in the current drawing.
BmShow - shows a previously hidden mechanical component or subcomponent in the current drawing.

BmXConvert

Converts X-Hardware solids in the current drawing to mechanical components.

Accessing the Command

command bar: *bmxconvert*

: ***bmxconvert***
Reports you in the command bar:
Number of X-Hardware solids converted to mechanical components: #

Command Options

No options.

Procedures

To convert geometry to a valid sheet metal part

Related Commands

BmForm - creates a new mechanical component and inserts it into the current drawing.
BmHardware - inserts a standard hardware part as a mechanical component in the current drawing.

MechanicalBrowserClose

Closes the Mechanical Browser panel.

Accessing the Command

command bar: *mechanicalbrowserclose*

: ***mechanicalbrowserclose***

Displays no prompts you in the command bar:

Command Options

No command options.

Related Commands

MechanicalBrowserOpen - opens the Mechanical Browser panel.
-ToolPanel - controls the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

MechanicalBrowserOpen

Opens the Mechanical Browser panel.

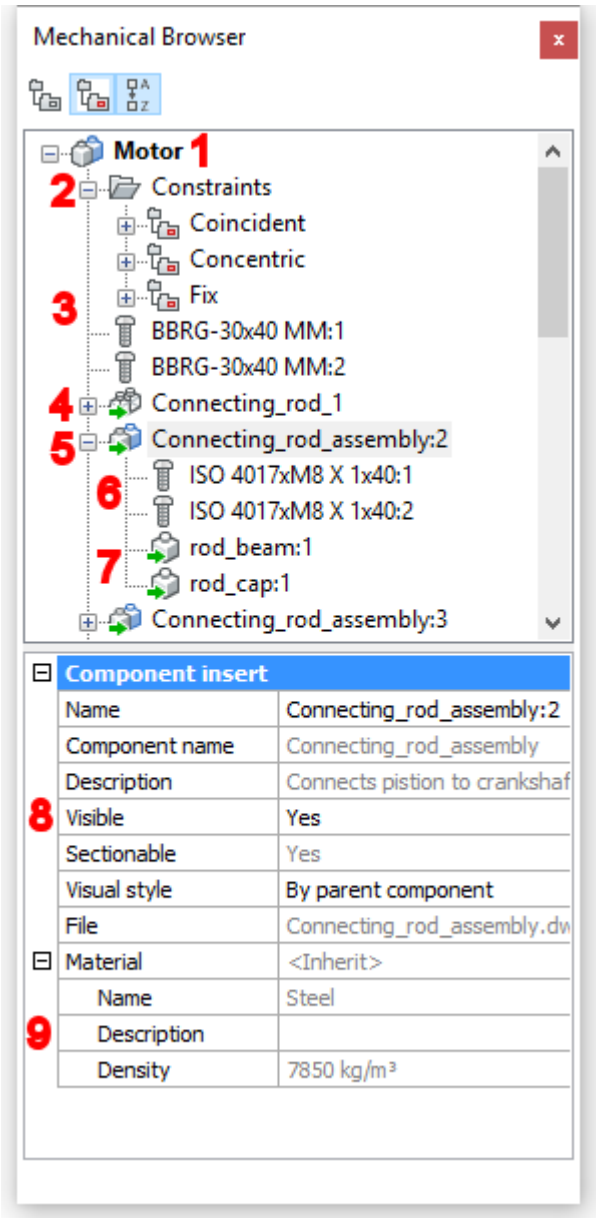
Accessing the Command

command bar: *mechanicalbrowseropen*
menu bar: *Assembly | Mechanical Browser*
ribbon: *Assembly | Tools | Mechanical Browser*
toolbar: *Assembly |* and *Parameters |*
context menu*: *Mechanical Browser*

* Place the cursor over a toolbar or a ribbon panel, then right click.

: ***mechanicalbrowseropen***

Displays a dialog box:



The Mechanical Browser allows navigation on the hierarchy of mechanical components for the current drawing and calling different commands for the inserts.

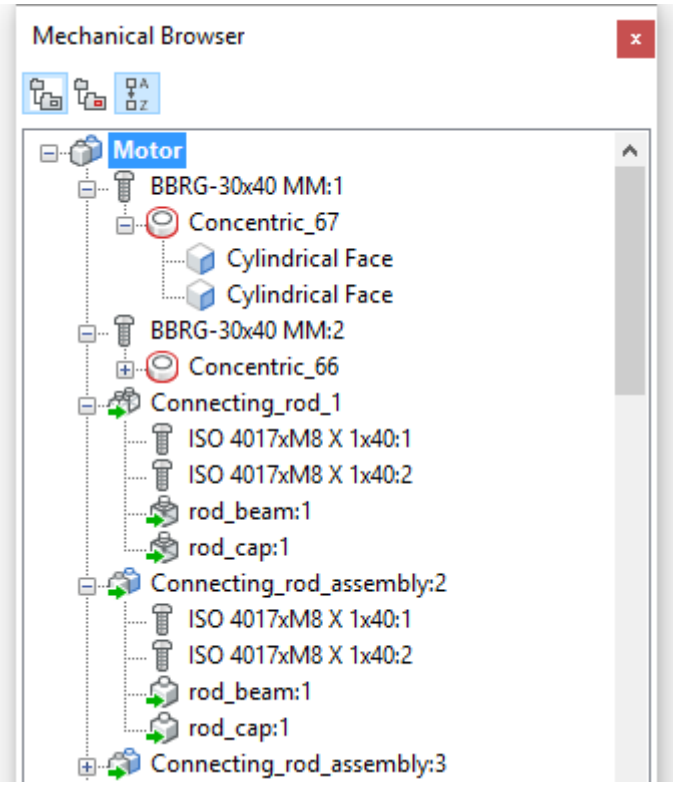
1. Component Name
2. 3D Constraints
3. Standard Part Component Insert (= block reference)
4. Collapsed Component Insert (= external reference)
5. Expanded Component Insert (= external reference)
6. Standard Part Subcomponent Insert (= nested block reference)
7. Subcomponent Insert (= nested external reference)
8. Properties of the selected item
    - *Name*: name of the component in the assembly (= internal name)
    - *Component name*: name of the component in the source file (= external name)
    - *Description*: description of the component
    - *Visible*: controls the visibility of the component in the drawing
    - *Sectionable*: controls whether the item is affected by the ViewSection command.

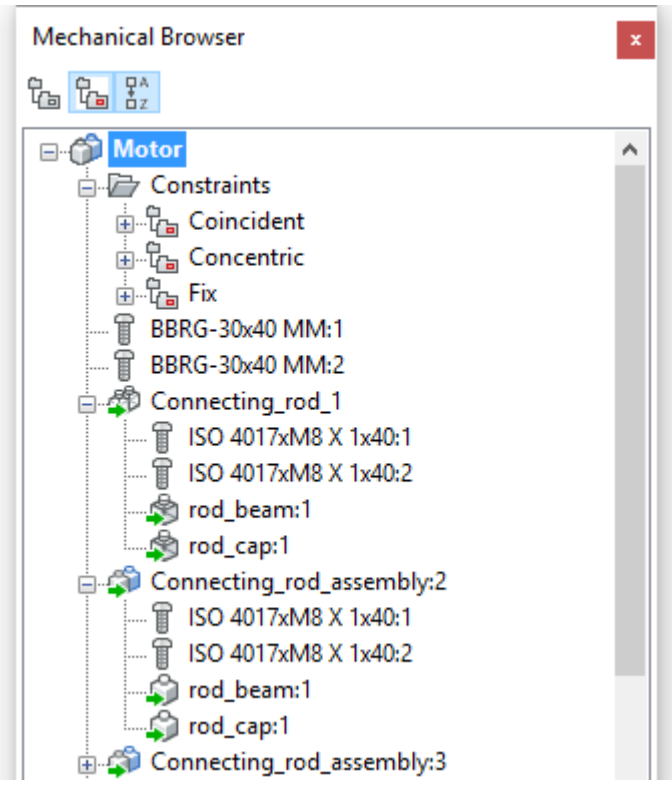
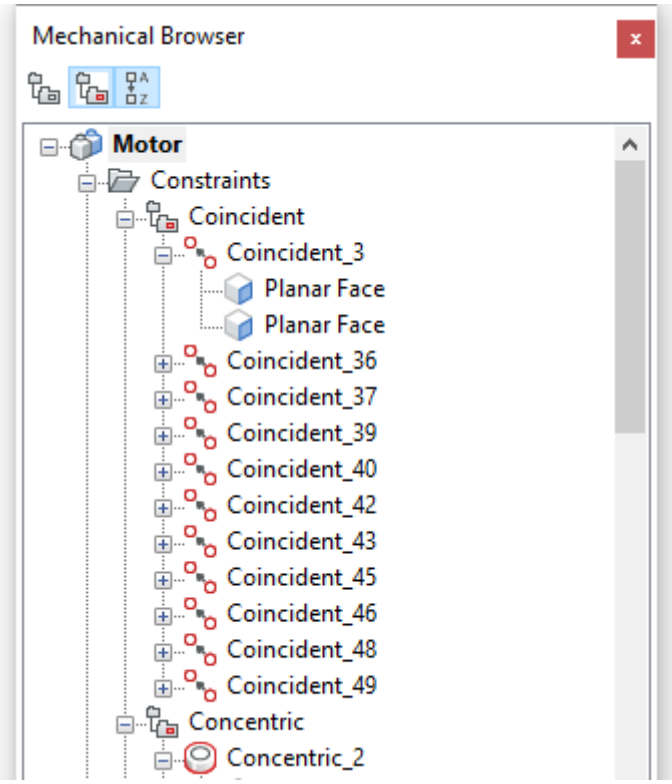
- *Visual Style*: defines the visual style of the component (see also BmVStyle command).
- *File*: source file of the component
- *Material*: Material of the selected item, as defined in the *Material* definition of the source file.
  If the *Material* definition of the source file is set to <Inherit>, the material is copied from the main component.

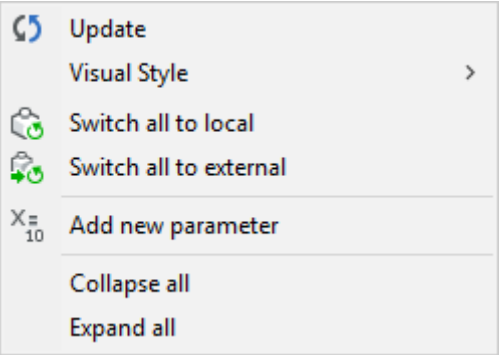
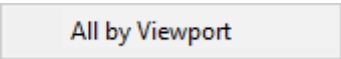
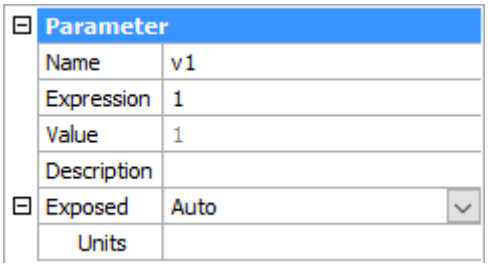
	Icons		
	Rendered visual style	Wireframe visual style	Hidden
Component that does not contain subcomponents	*	*	*
Component that contains subcomponents	*	*	*
Standard Part Component			

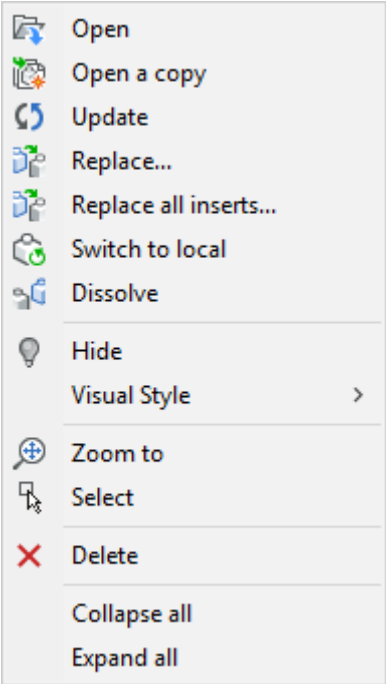
* External components have a green arrow added: or

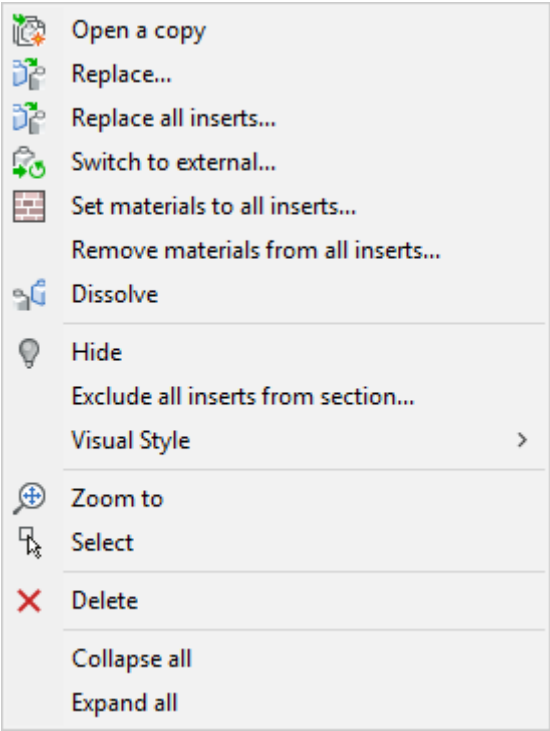
Command Options

Option	Description
Model	
Group by Entity	Groups 3d constraints by entity or mechanical component they are applied to. 

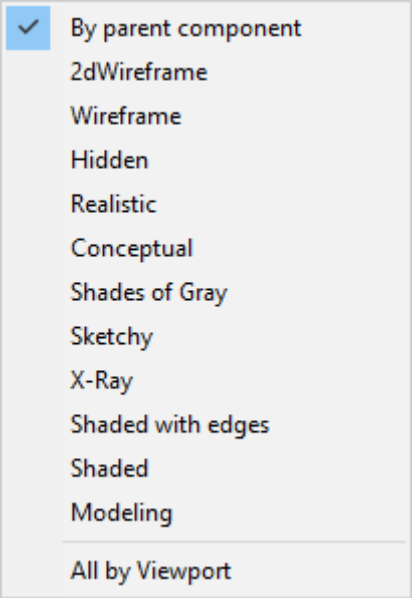
Group by Type	Groups mechanical components and 3d constraints in separate trees.  Groups 3d constraints by type. Expand a constraint to see the 3D features affected by the constraint.  C

Sort Alphabetically	When pressed ( ) the mechanical components and 3d constraints are sorted alphabetically, When depressed ( ) the mechanical components and 3d constraints are listed in the order they were added to the assembly.
Context Menus	
Main Component context menu	Right click the main component name:  • Update: Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified. • Visual Style: All by Viewport: Applies the current Visual Style to all components in the assembly (see the ShadeMode command).  • **Switch all to local**: Switches all components to internal components (see the BmLocalize command). • Switch all to external: Switches all components to external components (see the BmExternalize command). • Add new parameter: Creates a new parameter in the assembly.  Name: Identifies the parameter. The parameter name can be used in expressions of other parameters in the same component. Expression: Type a value or an expression. Value: Displays the current value of the parameter. Description: Optional description of the parameter. Exposed: Defines whether the parameter is visible and can be modified when the component is inserted in an assembly. Select the field, then click the down-arrow button and choose an option. A new parameter gets the *Auto* option by default. *Auto* = the parameter is exposed only if it does not depend on other parameters.

	ON = the parameter is always exposed. *OFF* = the parameter is never exposed. Units: Sets the dimension of the parameter: Linear, 2-dimensional or 3-dimensional. Select the field, then click the down-arrow button and choose an option. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.
Component context menu	Right click a component name. External components context menu (left), local components context menu (right): 

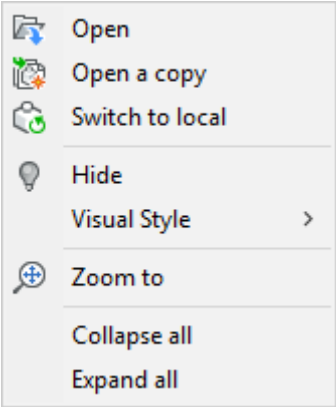
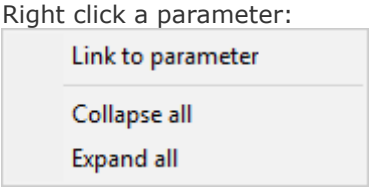


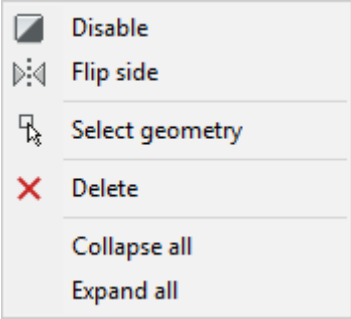
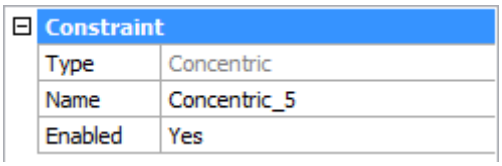
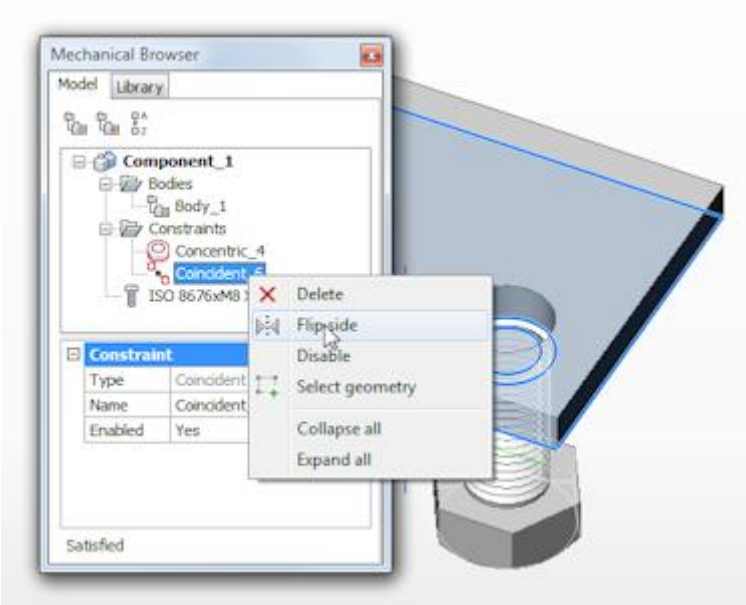
- Open: Opens the referenced drawing (see the BmOpen command).
- Open a copy: Opens a copy of a component insert as a new drawing (see the BmOpenCopy command).
- Update: Reloads all referenced components from external files and updates BOM tables (see the BmUpdate command).
- Replace: Replaces a component insert (see the BmReplace command). 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). Replacing local inserts turns them into external inserts.
- 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: Dissolves a mechanical component inserted in the current drawing (See the BmDissolve command).
- Hide ( ) / Show ( ): Hides or shows the selected component.
- Visual Style: Displays the *Visual Style* menu. Visual Styles that are saved in the current drawing are available.

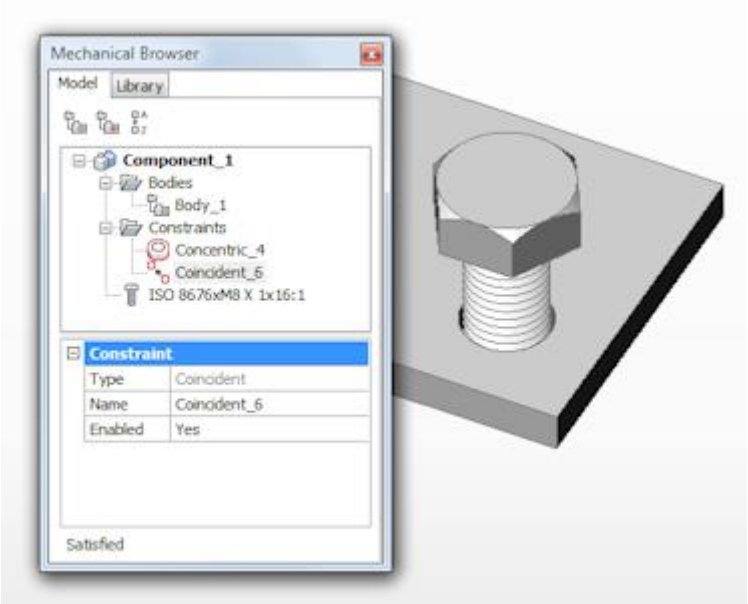


- By Viewport: Renders the selected component according to the current viewport shademode (see the ShadeMode command).
- All by Viewport: Renders all components according to the current viewport shademode (see the ShadeMode command).

- Zoom to: Zooms to the selection set.
- Select: Selects the component.
- Set materials to all inserts...: Assigns a physical material to all similar local components.
- Removes materials from all inserts...: Removes the physical material definition from all similar local components.
- Exclude all inserts from section: Sets the *Sectionable* property of all similar inserts to *NO*. Defines whether an insert is affected by the ViewSection command.
- Delete: Removes the selected component and its subcomponents from the assembly.
- Collapse all: collapses the main component and all components and subcomponents.
- Expand all: expands the main component and all components and subcomponents.

Subcomponent context menu	Right click a component name:  • 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). • Switch to local: Switches an external component to an internal component (see the BmLocalize command). • Hide ( ) / Show ( ): Hides or shows the selected subcomponent. • Visual Style: See Component Visual Style. • Zoom to: Zooms to the selection set. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.
Component parameter context menu	Right click a parameter:  • Link to parameter: Establishes a link between one or more component parameters and assembly parameters. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.

Constraint context menu	Right click a constraint:  • Disable/Enable: Toggles the selected constraint ON/OFF. The *Enabled* field in the Properties grid is adjusted accordingly.  • Flip side: Allows to change the relative position of an entity (vectors normal of the selected faces point in the same direction or in the opposite direction). This option can be applied to *Parallel*, *Coincident* and *Concentric* constraints only and on condition the constraint is applied to faces. 



-
- Select geometry: Selects the geometry, that is affected by the constraint, in the drawing.
- Delete: Deletes the selected constraint.
- Collapse all: collapses the main component and all components and subcomponents.
- Expand all: expands the main component and all components and subcomponents.

Properties	

Main component properties	Click the main component name:
	⊟ **Component**
	Name: Motor
	Description:
	File: D:\Bricsys Reports\V14\BIC2012-MCA
	Extension type:
	Insert as: External component
	Sectionable: Yes
	⊟ Material: Steel
	Name: Steel
	Description:
	Density: 7850 kg/m³
	• Name: Type a name for the main component. By default the name of the main component equals the file name.
	• Description: Optional description of the main component.
	• File: Shows the path and name of the dwg.
	• Extension type: Allows to identify the component as a BIM Component or a Sheet Metal Feature.
	• Insert as: Select either Internal Component or External Component.
	• Sectionable: Defines whether the component is affected by the ViewSection command.
	• Material: Material of the main component.
	The *Name*, *Density* and the optional *Description* are defined in the *Physical Materials* dialog box.
	Select the *Material* node, then do one of the following:
	  • Click the *Browse* button (...) to open the *Physical Materials* dialog and choose a material in the *Project* or *Library* list.
	  The *Density* property of the selected material is used by the BmMassProp command.
	  • Click the *Delete* button (x). The *Material* field reads *<Inherit>*.
	  When the component is inserted in an assembly, the *Material* definition is copied from the main component of the assembly.

Component insert properties	Click the component name: ⊟ **Component insert** Name: Connecting_rod_assembly:3 Component name: Connecting_rod_assembly Description: Connects pistion to crankshaft Visible: Yes Sectionable: Yes Visual style: By parent component File: Connecting_rod_assembly.dwg ⊟ Material: <Inherit> Name: Steel Description: Density: 7850 kg/m³ • Name: Type a name for the component. By default the name of the component equals the source file name. • Component name: Name of the component as defined in the source file. • Description: Description of the component as defined in the source file. • Visible: Allows to show or hide the component in the assembly. Click the *Visible* field, then click the down-arrow button and choose *Yes* or *No*. or Double click the *Visible* field to toggle the visibility of the component. • Sectionable: Defines whether the component is affected by the ViewSection command. • Visual stye: Allows to control the visual style for the component. By default the visual style is inherited from the viewport. Click the Visual style field, then click the down-arrow button and choose a visual style in the list. • File: Shows the path and name of the referenced dwg. • Material: Shows the material of the component. If the in material the source file is set to <Inherit>, the material is copied from the main component. If the material is defined explicitly in the source file, this material is used.

Subcomponent properties	Click the subcomponent name: ⊟ **Component insert** Name: rod_beam:1 Component name: rod_beam Description: Visible: Yes Sectionable: Yes Visual style: By parent component File: rod_beam.dwg ⊟ Material: <Inherit> Name: Steel Description: Density: 7850 kg/m³ • Name: Name of the component as defined in the source file of the parent component. • Component name: Name of the component as defined in the source file. • Description: Description of the component as defined in the source file. • Visible: Allows to show or hide the component in the assembly. Click the *Visible* field, then click the down-arrow button and choose *Yes* or *No*. or Double click the *Visible* field to toggle the visibility of the component. • Sectionable: Defines whether the component is affected by the ViewSection command. • Visual stye: Allows to control the visual style for the component. By default the visual style is inherited from the viewport. Click the Visual style field, then click the down-arrow button and choose a visual style in the list. • File: Shows the path and name of the referenced dwg. • Material: Shows the material of the subcomponent. If the in material the source file is set to <Inherit>, the material is copied from the main component or the parent component. If the material is defined explicitly in the source file, this material is used.
Selection Methods	
Select a component in the model.	The component highlights in the Mechanical Browser.
Click a component in the Mechanical Browser	The component highlights in the drawing.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

3dCompare - executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel.
BmDependencies - lists all files, containing component definitions inserted in the assembly, in the command window.
BmExternalize - switches internal components to external components.
BmHardware - inserts a standard hardware part as a mechanical component in the current drawing.
BmLocalize - switches external components to internal components.
BmOpen - opens the component insert drawing.
BmOpenCopy - opens a copy of a component insert as a new drawing.
BmReplace - replaces a component insert.
BmUpdate - reloads all referenced components from external files and updates BOM tables.
DesignTable - reates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser and parametric component properties.
-DesignTableEdit - allows to edit designtables.
MechanicalBrowserClose - closes the Mechanical Browser panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.

ReportPanelClose

Closes the Report panel.

Accessing the Command

command bar: *reportpanelclose*
context menu*: *Report*

* Place the cursor over a toolbar or a ribbon panel, then right click.

: ***reportpanelclose***
Displays no prompts in the command bar:

Command Options

No command options.

Related Commands

ReportPanelOpen - opens the Report panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

ReportPanelOpen

Opens the Report panel.

Accessing the Command

command bar: *reportpanelopen*
context menu*: *Report*

* Place the cursor over a toolbar or a ribbon panel, then right click.

: ***reportpanelopen***
Displays a dialog box.

Some commands use the *Report* panel to report the command output; e.g. all SheetMetal (Sm*) commands, DmAudit and DmStitch.
The *Report* panel is used instead of the command bar only if the REPORTPANEL system variable is ON.

Command Options

No command options.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

ReportPanelClose - closes the Report panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

StandardPartsPanelclose

Closes the Standard Parts panel.

Accessing the Command

command bar: *standardpartspanelclose*

: ***standardpartspanelclose***
Displays no prompts in the command bar.

Command Options

No command options.

Related Commands

StandardPartsPanelOpen - opens the Standard Parts panel, allowing to insert a standard hardware part as a mechanical component in the current drawing.
-ToolPanel - controls the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

StandardPartsPanelOpen

Opens the Standard Parts panel, allowing to insert a standard hardware part as a mechanical component in the current drawing.

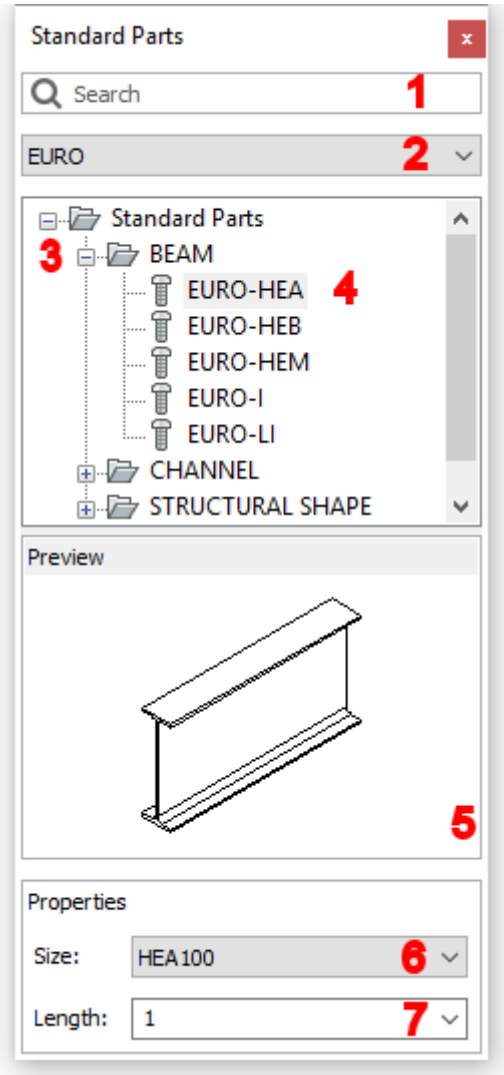
Accessing the Command

command bar: *standardpartspanelopen*
context menu*: *Standard Parts*

* Place the cursor over a toolbar or a ribbon panel, then right click.

: ***standardpartspanelopen***

Displays the Standard Parts panel:



1. Search field: type a search string then press Enter to search for a standard part in the currently selected group.
2. Groups button: click to select a group.
3. Standard part category (expanded)
4. Standard part types: click to select.
5. Preview.
6. Size list button: click to select a size.

7. Length field: choose a default length or type a length in the settings field.

To place a standard part in the current drawing:

1. Choose a group.
2. Choose a category.
3. Select a type.
4. Select a size in the list
5. Type the desired length in the Length field.
6. Drag the part from the expanded type list into the drawing.

Command Options

No command options.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

-BmHardware - inserts a standard hardware part as a mechanical component in the current drawing. Prompts you in the command bar.
BmInsert - inserts an existing mechanical component into the current drawing.
BmXConvert - converts X-Hardware solids in the current drawing to mechanical components.
MechanicalBrowserOpen - opens the Mechanical Browser panel.
StandardPartsPanelclose - closes the Standard Parts panel.
-ToolPanel - controls the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

N

Navigate

Allows to walk or fly through models using the well-known approach used in computer games.

Accessing the Command

command bar: *navigate*
menu bar: *View* | *Navigate*
toolbar: *View* |
Quad: *General* |

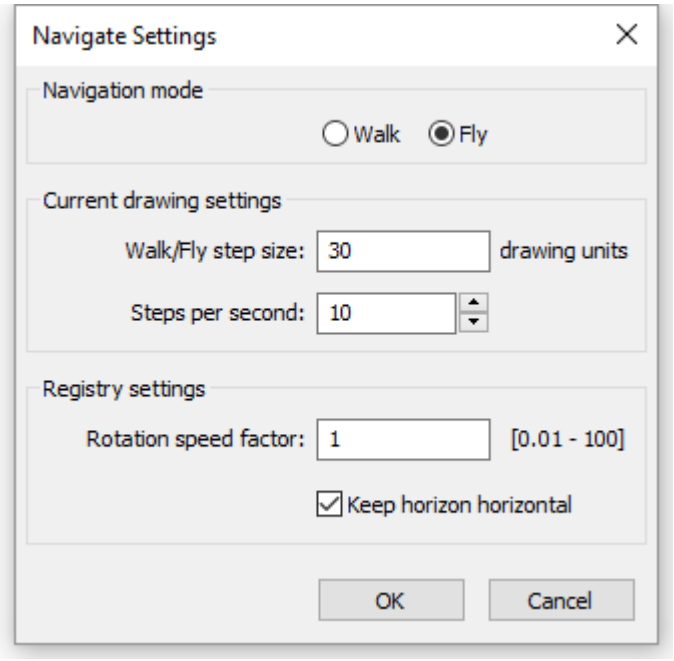
: ***navigate***
Prompts you in the command bar:
Flight mode is Off, STEPSIZE=30, STEPSPERSEC=10
>>Press Esc to quit or right click to modify navigation settings.

Hover the mouse cursor over the target location and hold down a navigation key as indicated in the table below. Move the cursor to change the target location.

Command Options

Keyboard		Description
A	Left Arrow	Move to the left
W	Up Arrow	Move forward
S	Down Arrow	Move backward
D	Right Arrow	Move to the right
F		Toggles Flight Mode ON/OFF. Flight mode OFF (= Walk Mode) restricts the movements to the XY-plane.
Esc (Escape)		Exits the navigate command.

Mouse	Description
Hold down the left button and drag	Look around.
Right click	Brings up the Navigation Settings dialog.



The number of steps is saved in the drawing through the STEPSPERSEC system variable.

The size of the steps is saved in the drawing through the STEPSIZE system variable.

The rotation speed is saved in the registry through the RTRotationSpeedFactor user preference.

Related Commands

RtRot - rotates 3D drawings in real-time.
RtWalk - walk left / right and forward / backward in a 3D scene.

Nearest

Toggles Nearest entity snap mode; snaps to the nearest geometry on entities.

Accessing the Command

command bar: *nearest*
transparent: *nea*
toolbar: *Entity snaps* |

: ***nearest***

Displays no prompts in the command bar.

When nearest entity snap is on, the cursor snaps to the nearest geometry.

This entity snap works with 3D solid models.

Command Options

Option	Description
Nearest	Entering the command toggles Nearest entity snap. It snaps to the nearest geometry on entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.

Netload

Loads a .NET application.

Accessing the Command

command bar: *netload*

: ***netload***

If FILEDIA is ON, a standard file selection dialog box displays.

Select the application to be loaded and press the OK button.

If FILEDIA is OFF, Prompts you in the command bar:
Assembly file name: type the name of the application to be loaded in the command bar, then press Enter.

Command Options

No command options

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
Appload - loads DRX, LISP, and SDS applications to run inside BricsCAD.
Commands - lists the names of internal and external commands.
CuiLoad and CuiUnload - loads and unloads menu files.
Customize - customizes the user interface.
MenuLoad and MenuUnload - loads and unloads menu files.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaLoad and -VbaLoad - loads VBA projects.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

New

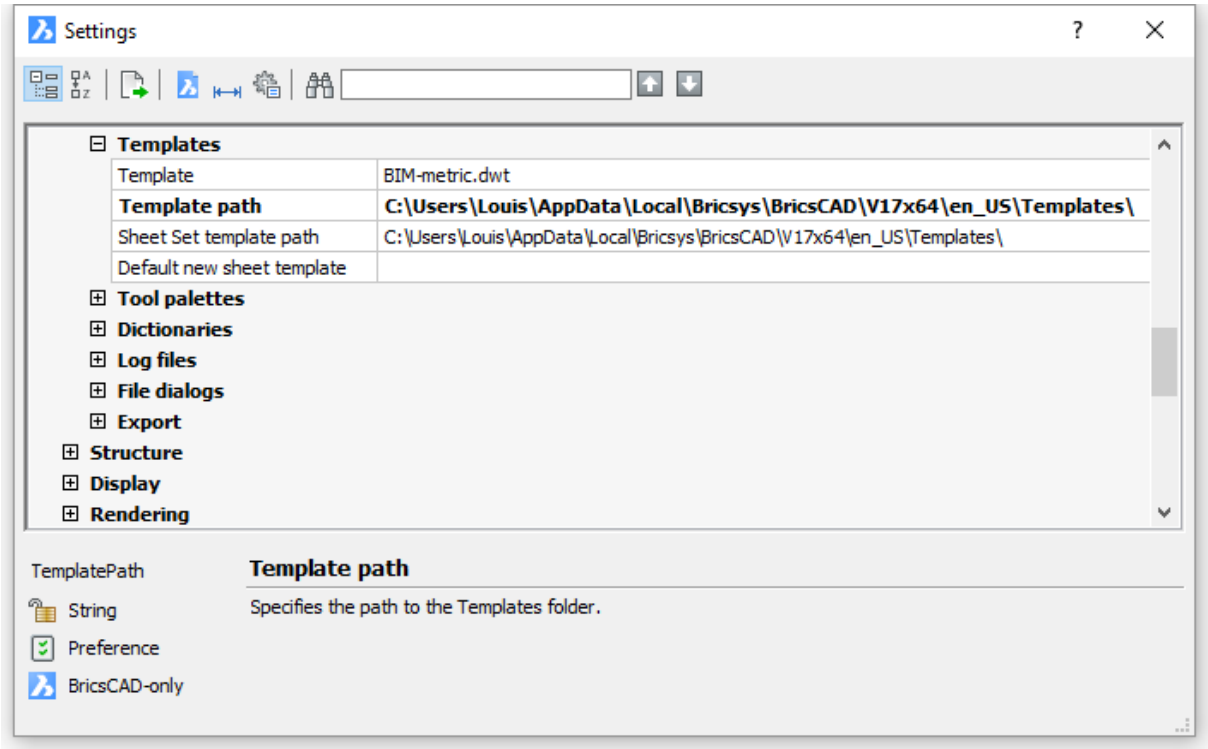
Starts new drawings.

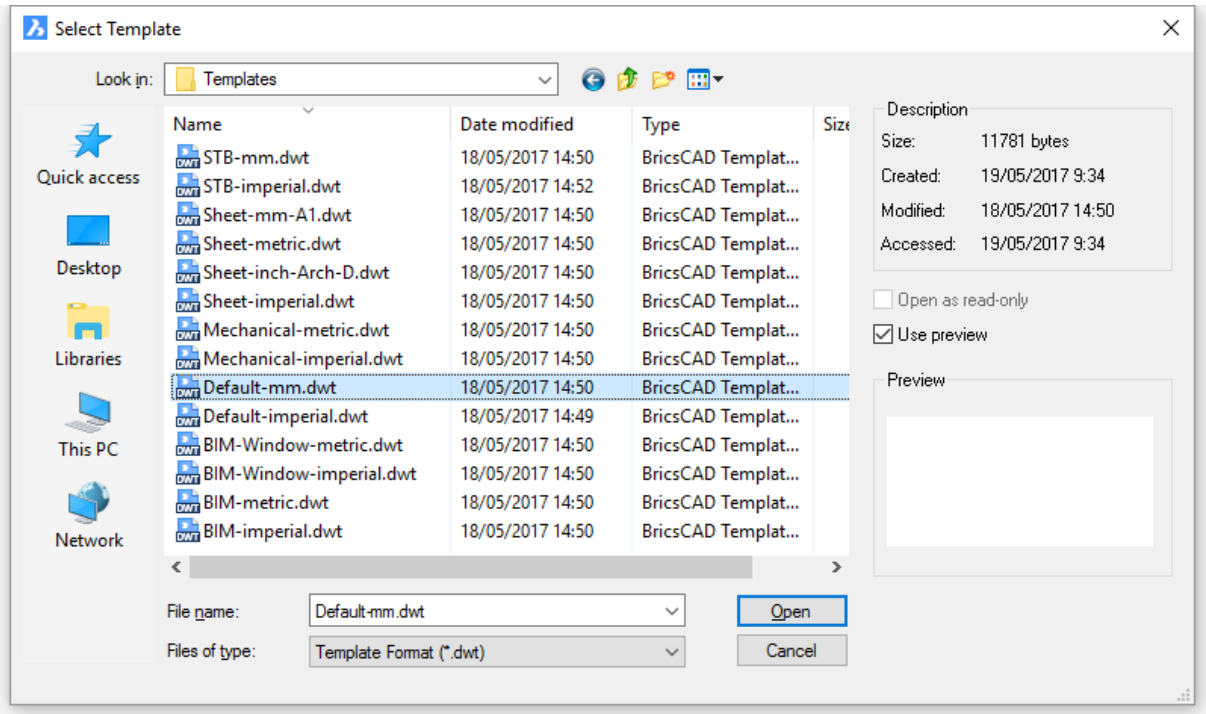
Accessing the Command

command bar: *new*
shortcut: *Ctrl+N*
menu bar: *File | New*

: ***new***

Opens Select Template dialog. The default template folder is defined by the TemplatePath user preference:





Choose a DWT template file or DWG drawing file, and then click **Open**.

- DWT template files define the properties of new drawings.
- DWG drawing files are opened like the **Open** command.

*When the **StartUp** system variable is turned on, this command displays the same dialog box as the **NewWiz** command*

Command Options

Option	Description
File name	Specifies the file name. To open more than one file at a time, hold down the **Ctrl** key while selecting file names.
Files of type: Template Format (*.dwt) / Template Format (*.dwt) / Standard Drawing File (*.dwg)	Chooses the type of drawing file: • **DWT** - specifies template files. • **DWG** - specifies drawing files.
Open as read-only	Toggles read-only status of drawings: • **On** - opens drawings read-only; they can be edited but not saved to the same file name. • **Off** - open drawings normally. Use the **SaveAs** command to save by another name. Template drawings are always opened as read-only.

Use Preview	Toggles previews of the drawing: • **On** - displays preview image in Preview area. • **Off** - does not display preview image. Many template files are empty, and appear to not have a preview.
Open	Opens the file.

Procedures

Working with document tabs

Procedures

To create a drawing using a default template drawing
To set the default template drawing
To create a drawing from scratch
To set the Plot style policy
To create a drawing using a template drawing
To create a drawing using the New command
To set the Template Path variable
To create a drawing using a wizard

Related Commands

Units - sets units through the Settings dialog box.
-Units - sets units through the command bar.
NewWiz - sets units and other parameters for new drawings.
QNew - opens new drawings.

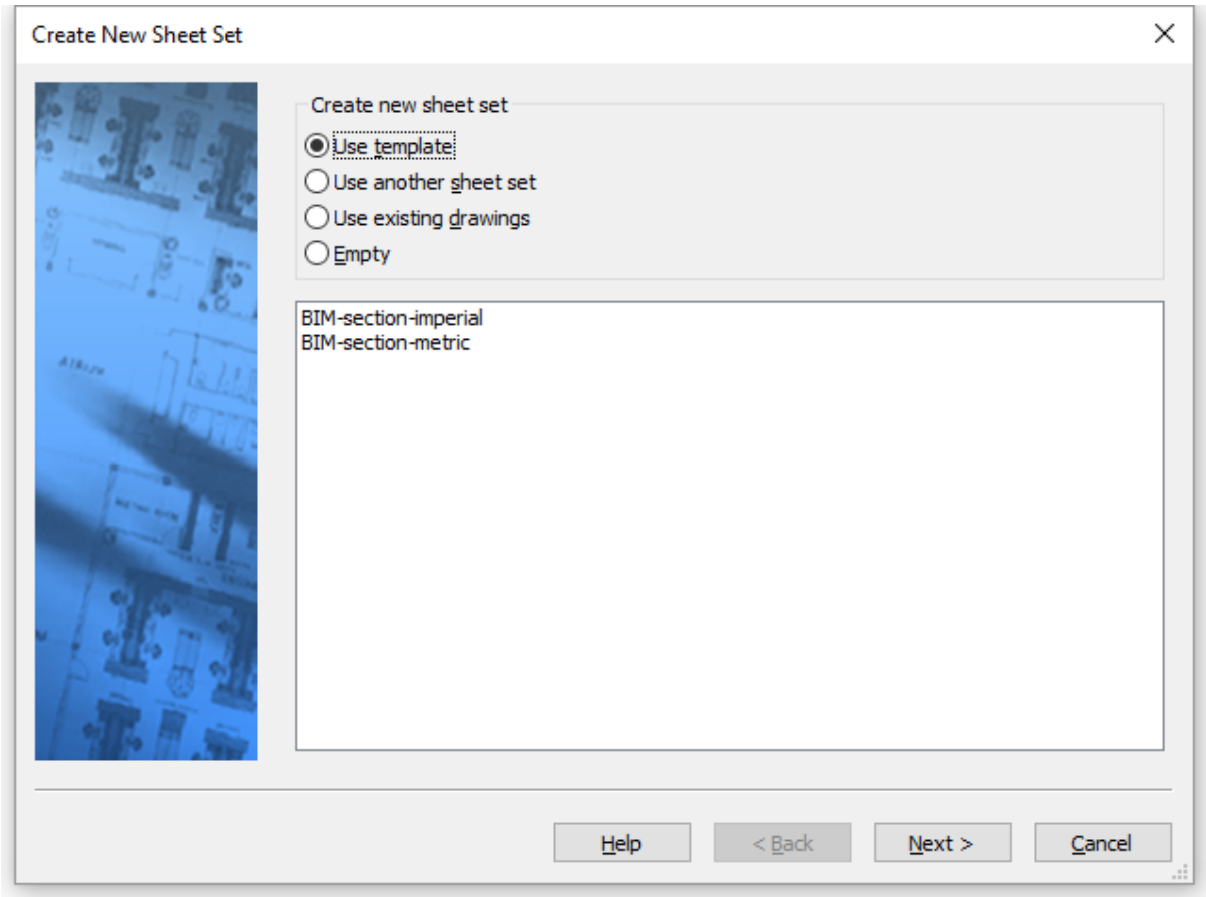
NewSheetSet

Creates a new sheet set.

Accessing the Command

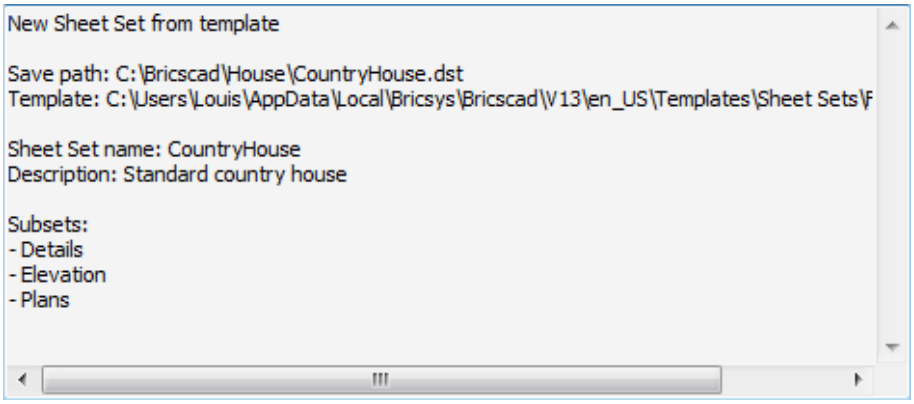
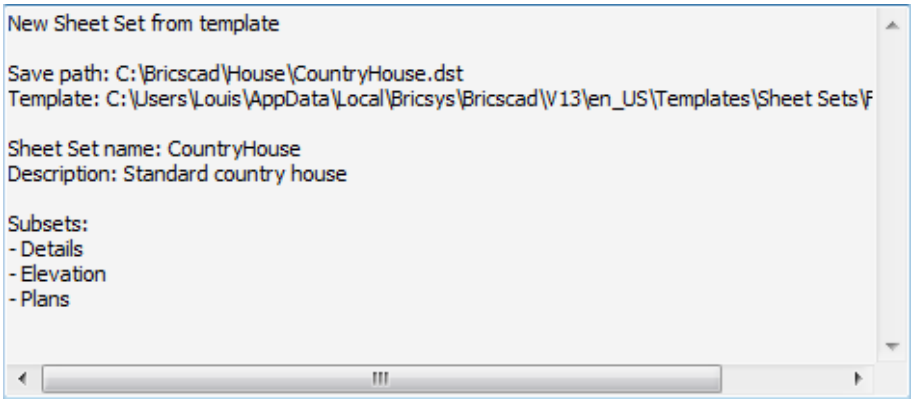
command bar: *newsheetset*
toolbar: *Drawing Explorer - Sheets* |

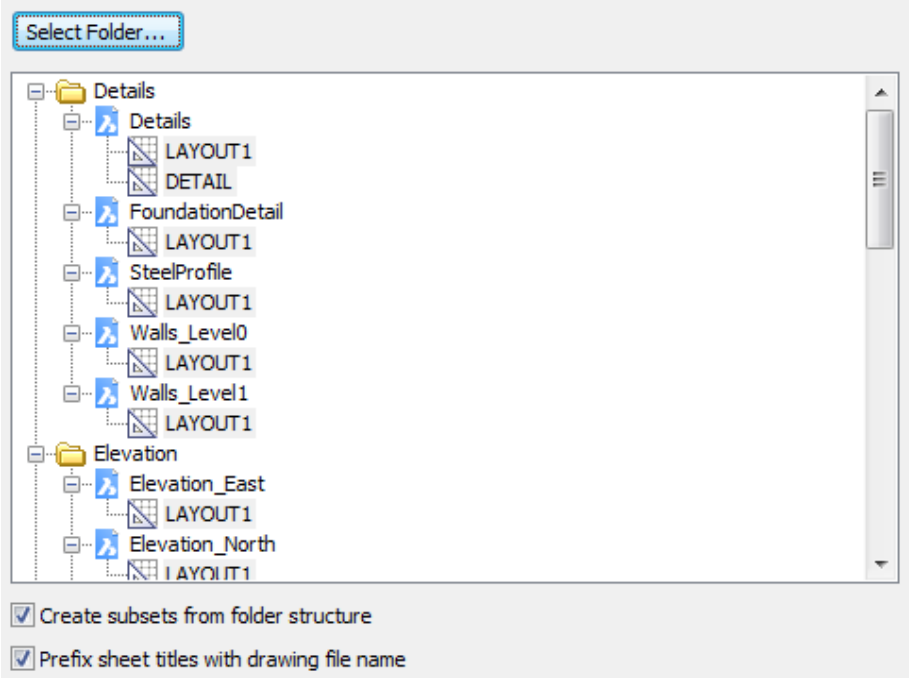
: ***newsheetset***
Displays a dialog box



Command Options

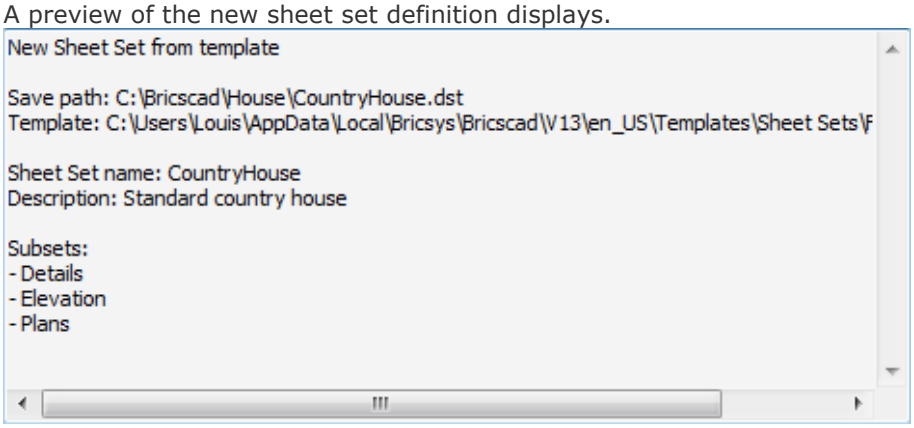
Option	Description
Use Template	Create a new sheet set using the settings in a template sheet set. Sheet set files (*.dst) in the *Sheet Sets Template* folder, as defined by the SheetSetTemplatePath user preference, are available. Default path: C:\Users\<UserName>\AppData\Local\Bricsys\BricsCAD\V[Version]\en_US\Templates\Sheet Sets 1. Choose a template sheet set. 2. Click the *Next* button. 3. Type a name in the *Save sheet set file as* field, then click the *Browse* button ( ).

	4. Select a folder, then click the *Save* button. 5. (option) Type a description in the *Description* field. 6. Click the *Next* button. 7. A preview of the new sheet set definition displays.  New Sheet Set from template Save path: C:\Bricscad\House\CountryHouse.dst Template: C:\Users\Louis\AppData\Local\Bricsys\Bricscad\V13\en_US\Templates\Sheet Sets\F Sheet Set name: CountryHouse Description: Standard country house Subsets: - Details - Elevation - Plans 8. Click the *Finish* button to create the new sheet set file.
Use another sheet set	Create a new sheet set using the settings in an existing sheet set. 1. Click the Browse button (...), then select a *.dst file. 2. Type a name in the *Save sheet set file as* field, then click the *Browse* button (...). 3. Select a folder, then click the *Save* button. 4. (option) Type a description in the *Description* field. 5. Click the *Next* button. 6. A preview of the new sheet set definition displays.  New Sheet Set from template Save path: C:\Bricscad\House\CountryHouse.dst Template: C:\Users\Louis\AppData\Local\Bricsys\Bricscad\V13\en_US\Templates\Sheet Sets\F Sheet Set name: CountryHouse Description: Standard country house Subsets: - Details - Elevation - Plans 7. Click the *Finish* button to create the new sheet set file.
Use existing drawings	Create a new sheet set from an existing set of drawings in a folder and its subfolders. 1. Select the folder which contains the drawings and drawing subfolders you want to include in the new sheet set. All layouts in the drawings in the selected folder and in its subfolders - in any - are selected.



2. Click the *Next* button.
3. Type a name in the *Save sheet set file as* field, then click the *Browse* button (...).
4. Select a folder, then click the *Save* button.
5. (option) Type a description in the *Description* field.
6. Specify the default drawing template for new sheets in the sheet set.

   Click the *Browse* button (...) to select a template drawing.
7. Click the *Next* button.
   A preview of the new sheet set definition displays.



8. Click the *Finish* button to create the new sheet set file.

Empty	Creates an empty sheet set. 1. Type a name in the *Save sheet set file as* field, then click the *Browse* button ( ). 2. Select a folder, then click the *Save* button. 3. (option) Type a description in the *Description* field. 4. Specify the default drawing template for new sheets in the sheet set. Click the *Browse* button ( ) to select a template drawing. 5. Click the *Next* button. A preview of the new sheet set definition displays. 6. Click the *Finish* button to create the new sheet set file.

Related Commands

SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.
OpenSheetSet - opens an existing sheet set.

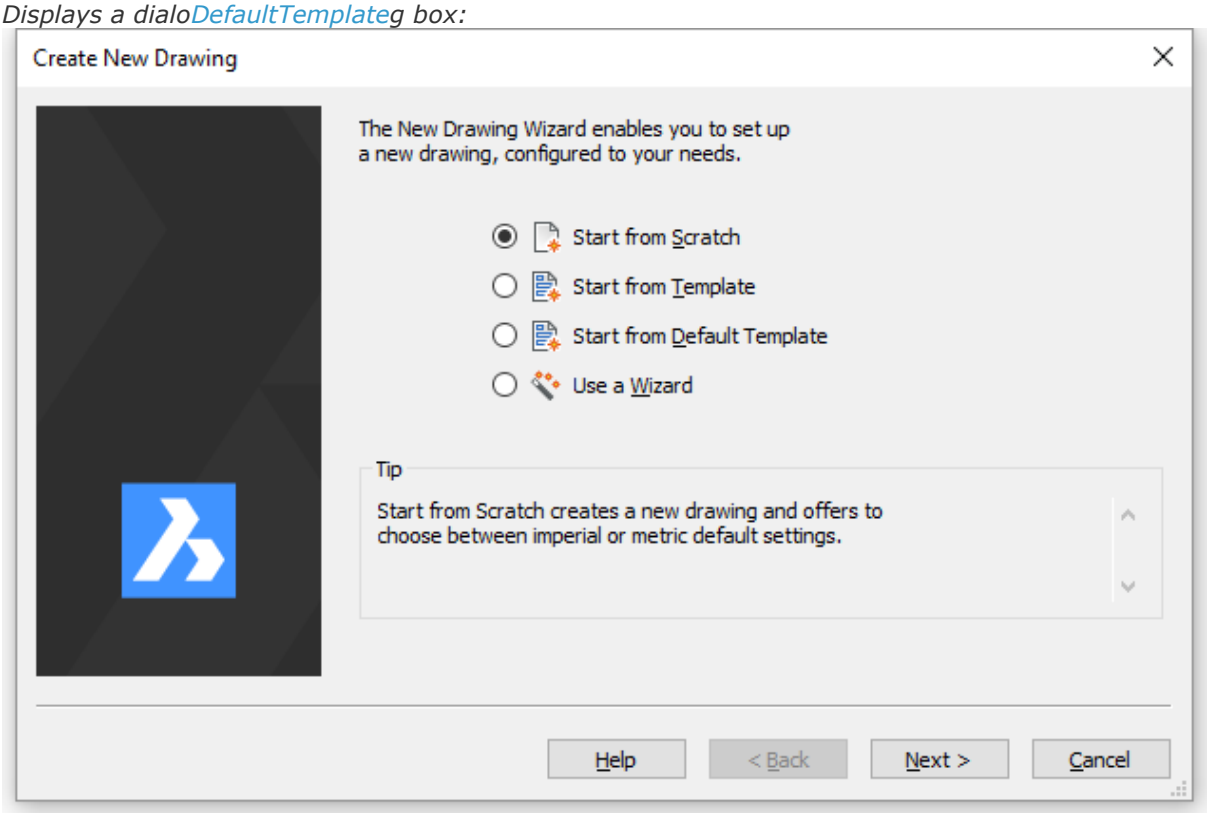
NewWiz

Starts new drawings with the *New Drawing Wizard*.

Accessing the Command

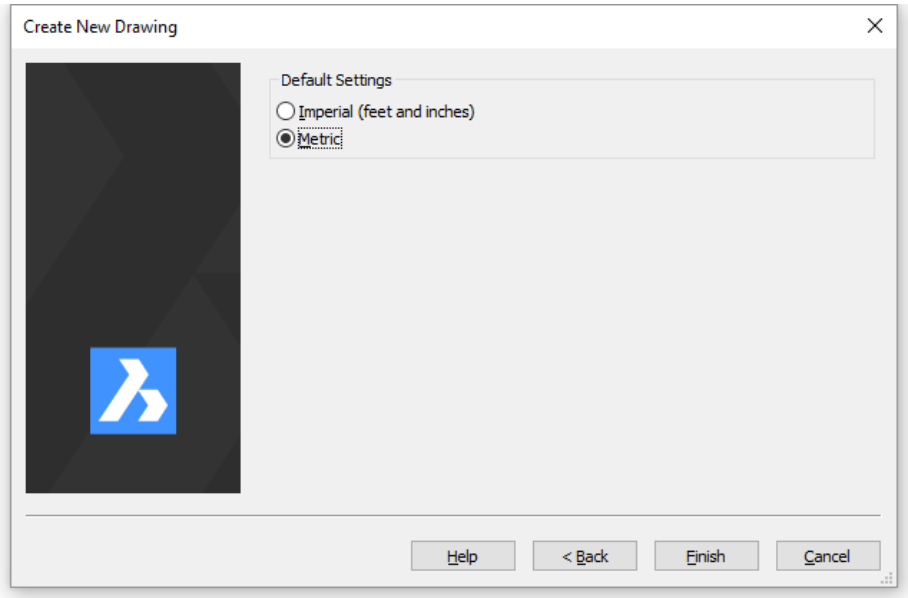
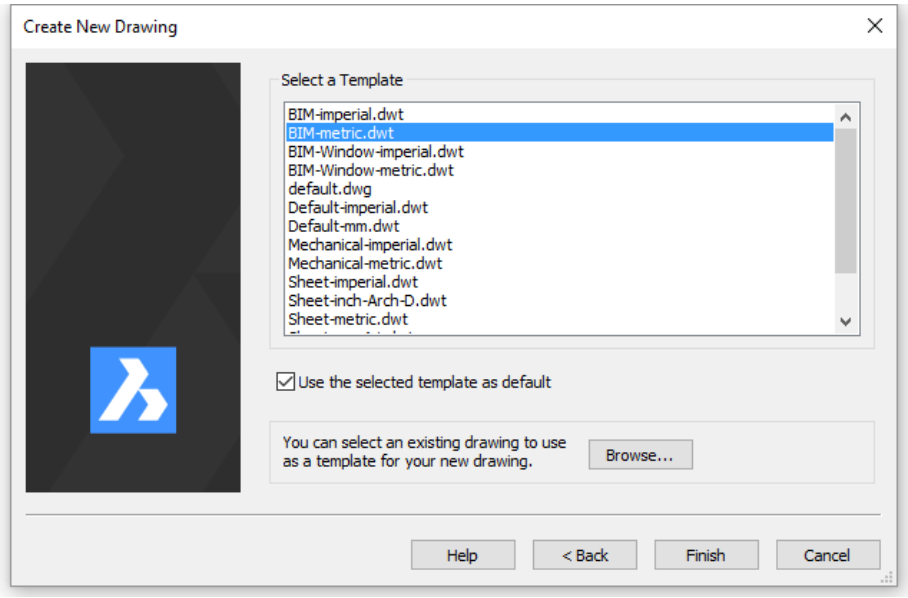
command bar: *newwiz*
alias: *ddnew*
menu bar: *File | New Wizard*

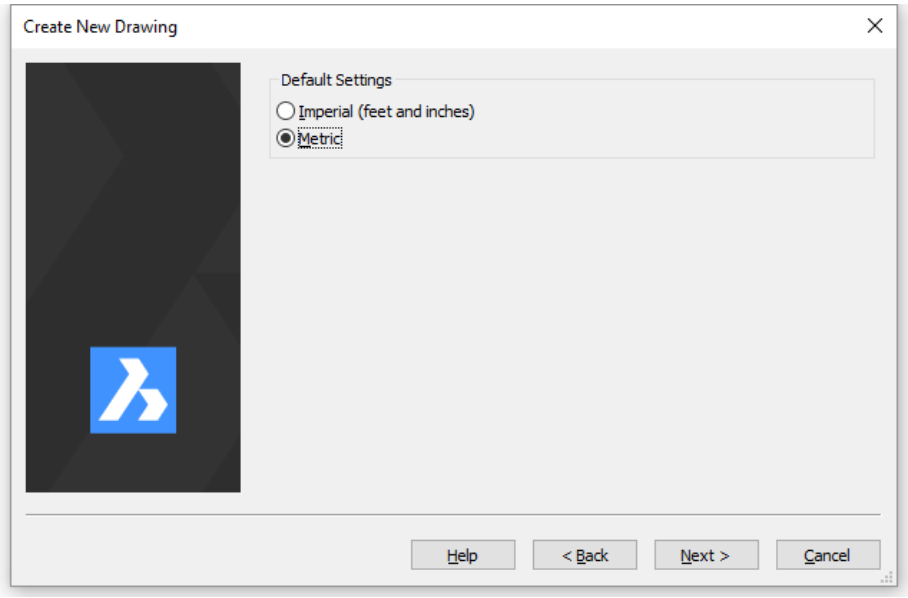
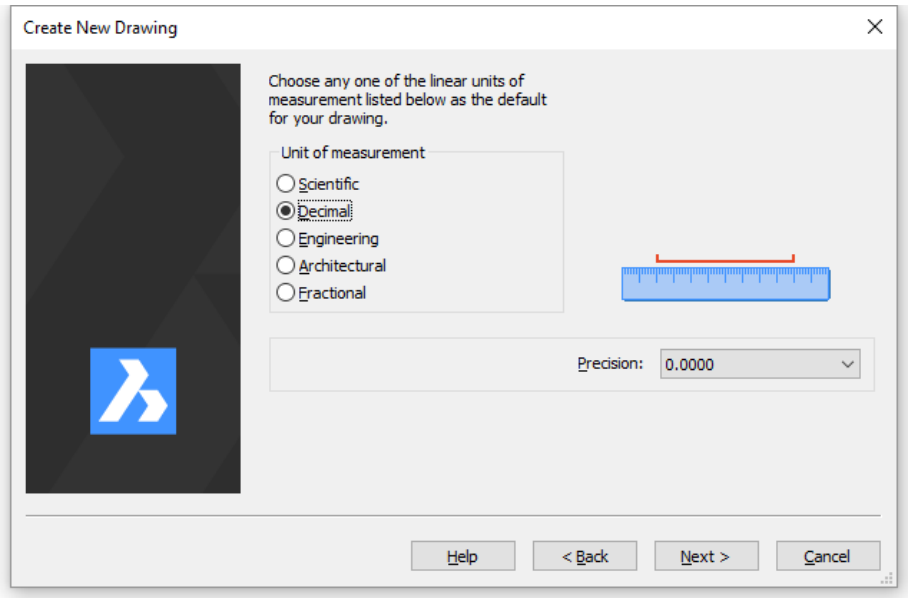
: ***newwiz***
Displays a dialoDefaultTemplateg box:



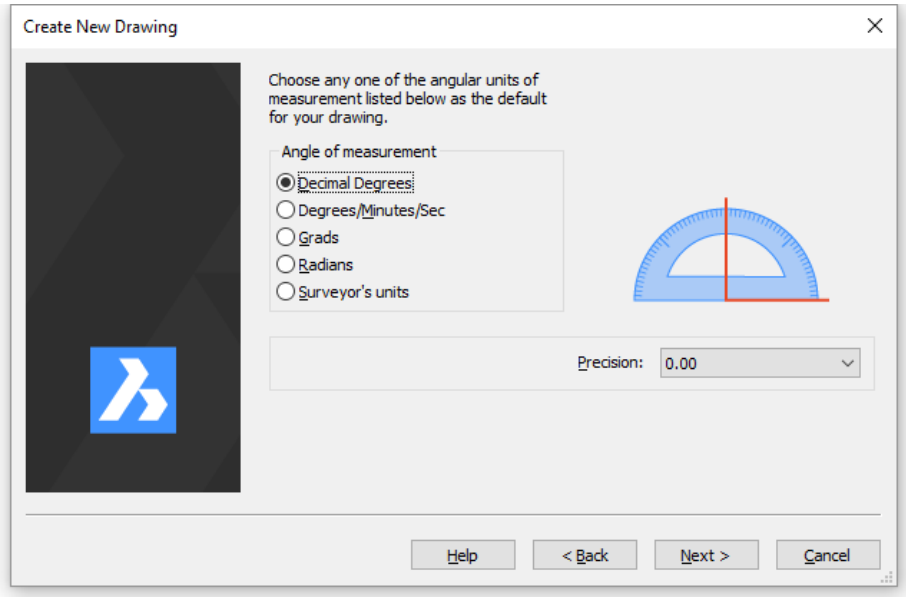
*Choose an option, and then click **Next**.*
Press ***F1*** to access help.

Command Options

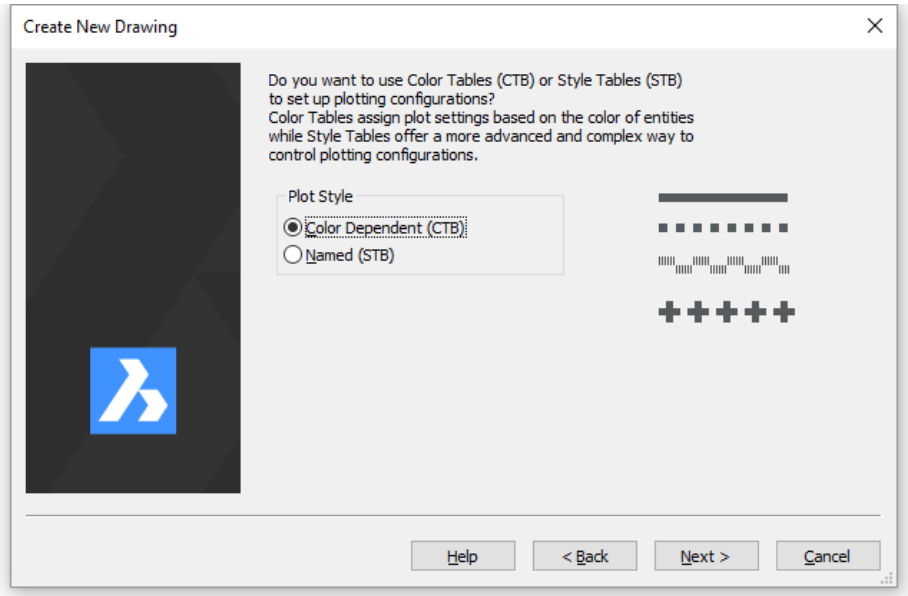
Option	Description
Start from Scratch	Starts new drawings using one of the default template files associated with these design context and measurement systems:  Choose a measurement system, and then click Finish: • Imperial (feet and inches) - choose the default-imperial.dwt file for imperial drawings. • Metric - choose the default-mm.dwt template file metric drawings.
Start from Template	Starts new drawings with a template file:  Choose a template file. Optionally check the *Use the selected template as default* option. Then click Finish. STB is short for "style table," which refers to template drawings with plot styles.

Start from Default Template	Starts new drawings with the default template as defined by the BASEFILE system variable.
Use a Wizard	Starts new drawings with a wizard:  Choose a measurement system, and then click Next: • Imperial (feet and inches) - choose the default-imperial.dwt file for imperial drawings. • Metric - choose the default-mm.dwt template file metric drawings.  Select a linear unit of measurement, then click Next:

1

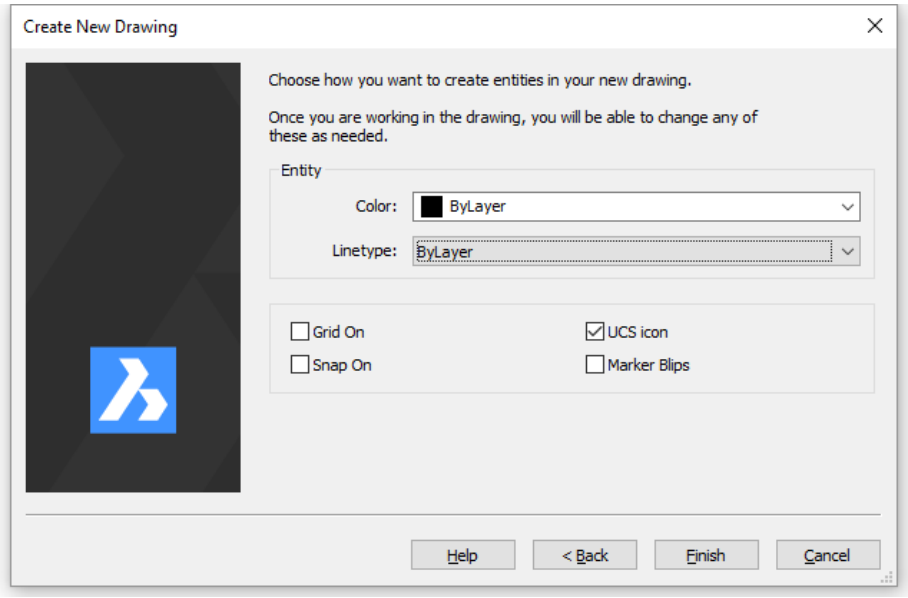


Choose an angular measurement unit, and then click Next:



Choose whether the drawing uses color-dependent plot styles or named plot styles, and then click Next:

- Color-dependent - uses plot styles that are simpler to employ, but less flexible.
- Named - uses plot styles are more flexible but more complex, because they can control all properties of every entity during plotting.



Choose the default working color and linetype, and then click Finish:

- Color - chooses ByLayer, which lets layers control the colors of entities. See Color and Layer commands.
- Linetype - chooses ByLayer, for the same reason. See Linetype and Layer commands.

Toggle the grid, snap, UCS icon, and marker blips:

- Grid On - turn on for seeing the drawing extents and relative distances. See Grid command.
- Snap On - turn on for setting the cursor increment. See Snap command.
- UCS icon - turn on for 3D drawings; turn off for 2D drawings. See UcsIcon command.
- Marker blips - turn off to keep screen clean. See Blipmode command.

Procedures

To create a drawing using a default template drawing
To set the default template drawing
To create a drawing from scratch
To set the Plot style policy
To create a drawing using a template drawing
To create a drawing using the New command
To set the Template Path variable
To create a drawing using a wizard

Related Commands

Units - sets units through the Settings dialog box.
-Units - sets units through the command bar.
New - starts new drawing in Imperial or metric units.
QNew - opens new drawings.

Node

Toggles Node entity snap mode; snaps to point entities.

Accessing the Command

command bar: *node*
transparent: *nod, poi*
toolbar: *Entity Snaps* |

: ***node***

Displays no prompts in the command bar.

When node entity snap is on, the cursor snaps to point entities.

Command Options

Option	Description
Node	Entering the command toggles Node entity snap. It snaps to point entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

None

Turns off all entity snap modes.

Accessing the Command

command bar: *none*
transparent: *non*
toolbar: *Entity Snaps* |

: ***none***
Displays no prompts in the command bar.

Command Options

Option	Description
None	Entering the command turns off all entity snap modes. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

O

ObjectScale

Adds or removes supported scales for annotative entities through a dialog box.

Accessing the Command

command bar: *objectscale*

properties bar: *Annotative Scale* | ...

: ***objectscale***

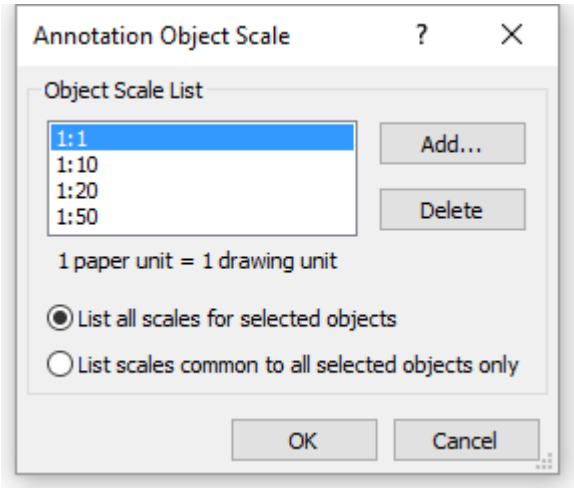
Prompts you in the command bar:

Select annotative entities: (Select entities)

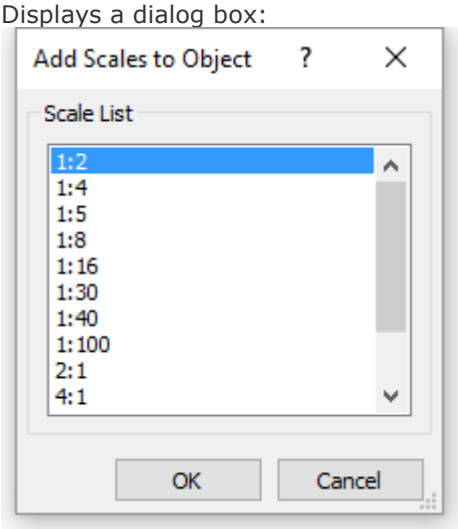
Entities in set: 2

Select annotative entities: (Select more entities or press Enter to continue.)

Displays a dialog box:



Command Options

Option	Description
Add	Adds annotation scales to the selected annotative entities. Displays a dialog box:  Select one or more scales in the list, then press the OK button to add the selected scales. Press and hold the Ctrl key to: • select multiple scales. • to unselect a scale To select a series of scales: select the first scale, then press and hold the Shift key and select the last scale.
Delete	Removes annotation scales from the selected annotative entities. Select the scale(s) to remove in the *Entity Scale list*, then press the *Delete* button. Press and hold the Ctrl key to: • select multiple scales. • to unselect a scale It is not possible to delete all scales. If you select all scales, the *Delete* option is not available. Choose *No* in the *Annotative* field on the Properties Bar to remove the annotative property.
List all scales for selected entities	The union of all scales for the selected entities are listed.
List scales common to all selected entities only	Only scales that are shared by all selected entities are listed.

Related Commands

-ObjectScale - adds or removes supported scales for annotative entities at the command line.
ScaleListEdit - edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport. Defines the available annotation scales for annotative entities.

-ObjectScale

Adds or removes supported scales for annotative entities at the command line.

Accessing the Command

command bar: *-objectscale*

: ***-objectscale***
Prompts you in the command bar:

Select annotative entities: **(Select entities)**

Entities in set: 1

Select annotative entities: *(Select more entities or right click to stop)*
Enter an option [Add/Delete/?] <Add>: (Enter an option)

Command Options

Option	Description
Add	Adds annotation scales to the selected annotative entities. Prompts you: Enter name scale to add or [**?**] <1:10>: Type a scale, then press Enter. x entities updated to support annotation scale '1:10'. Enter name scale to add or [?]: **Type another scale or press Enter to stop.**

Delete	Deletes annotation scales from the selected annotative entities.
	Prompts you:
	Enter name scale to delete or [?] <1:10>: Type a scale, then press Enter.
	x entities scale removed.
	Enter name scale to delete or [?]: Type another scale or press Enter to stop.
	
	Type ? to display a list of all scales of the selected objects.
	Press F2 to open the *Prompt History* window.
	
	`Scale Name  Paper Units  Drawing Units  Effective Scale`
	`----------  -----------  -------------  ---------------`
	`1: 1:1  1.0000  1.0000  1.0000`
	`2: 1:20  1.0000  20.0000  0.0500`
	`3: 1:50  1.0000  50.0000  0.0200`

?	Displays a list of available annotation scales as defined in the scale list (see the ScaleListEdit command). Press F2 to open the *Prompt History* window.

```
     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
```

Related Commands

ObjectScale - adds or removes supported scales for annotative entities through a dialog box.
ScaleListEdit - Edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport. Defines the available annotation scales for annotative entities.

Offset

Creates parallel copies of 2D linear entities, 3D solid faces and detected boundaries.

Accessing the Command

command bar: *offset*
alias: *o*
menu bar: *Modify | Offset*
toolbar: *Modify |*
Quad: *2D Editing |*

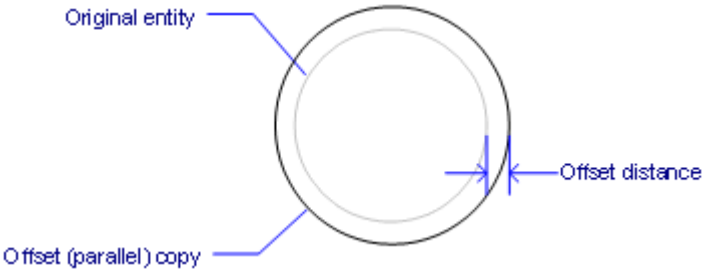
: ***offset***
Prompts you in the command bar:

Enter offset distance or [Through point/Erase/Layer] <20.0>: (Enter a distance, or choose an option.)

Select entity/subentity or [Exit] <Exit>: (Choose a single entity.)

Select side for parallel copy or [Both sides/Multiple]: (Pick a point or choose an option.)

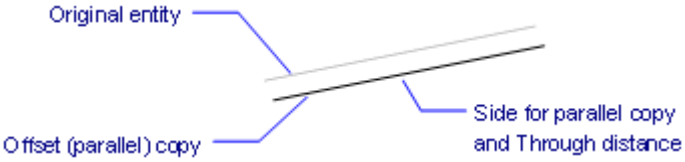
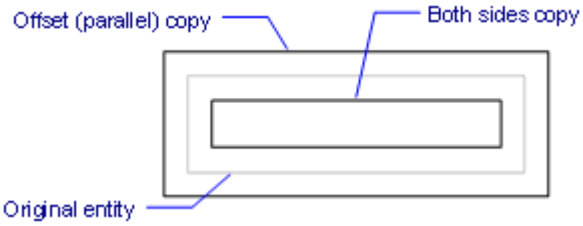
Select entity/subentity or [Undo/Exit] <Exit>: (Choose another entity, type U to undo the previous parallel copy or press Enter to exit this command.)

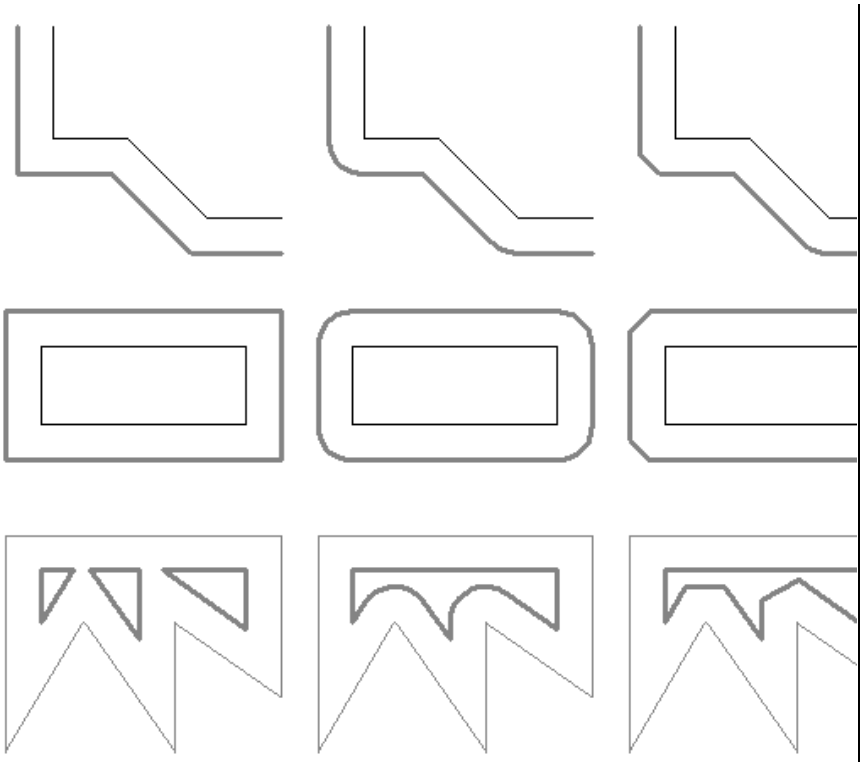
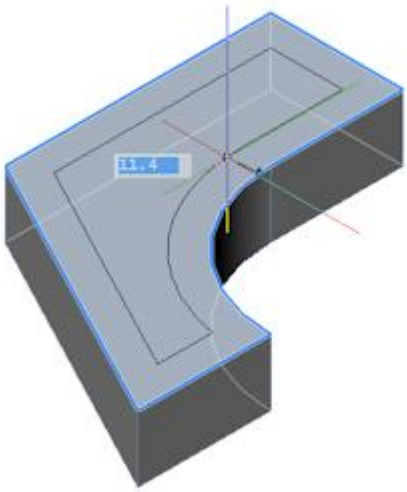


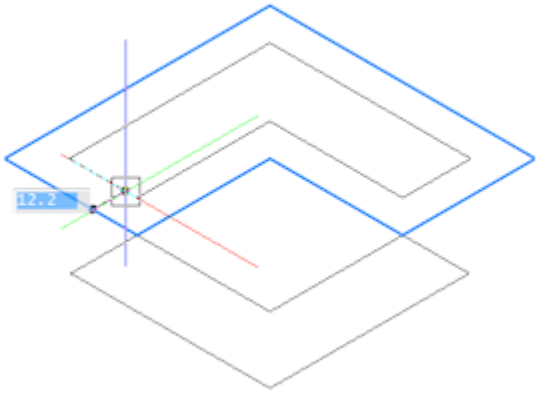
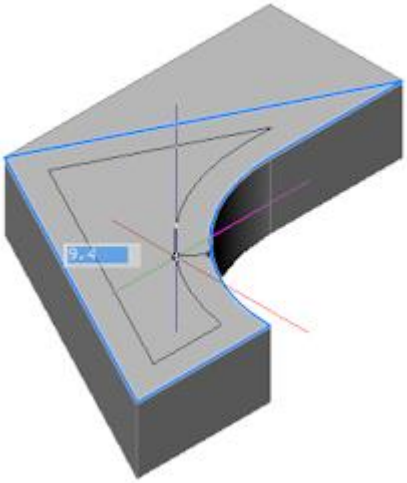
OFFSETGAPTYPE settings variable controls how potential gaps between segments are treated when polylines are offset.

Command Options

Option	Description
Distance	Specifies the distance between the original entity and its parallel copy. Do one of the following to specify the distance: • Press Enter to accept the current distance. • Type a value. • Specify two points in the drawing. The offset distance is saved to the *OFFSETDIST* system variable.
Erase	Deletes the source entity. This option sets the value of the *OFFSETERASE* system variable. The status of *OFFSETERASE* remembered for each open drawing separately. The value of *OFFSETERASE* is not saved with the drawing, but reset (Off) when you (re)open a drawing.
Through point	Specifies the offset distance by pointing.

	When you pick a point at the "Side for parallel copy" prompt, your pick point specifies the offset distance.
Select entity	Selects the entity of which to make the parallel copy. Only one entity can be selected; the command repeats automatically so that you can offset additional entities. Press Enter or Esc to exit the command.
Side for parallel copy	Specifies the side on which to make the offset (parallel) copy.  If the Through Point option was selected, your pick point also specifies the offset distance.
Both sides	Places offset copies on both sides of the entity. 
Layer	Select layer for offset entities? [Current/Source] <Source>: Choose an option: • Current: Create the offset on the current layer. • Source: Create the offset on the layer of the source entity.
Undo	Undoes the previous offset operation.
OFFSETGAPTYPE	The *OFFSETGAPTYPE* settings variable specifies how possible gaps are filled when offsetting polylines.

	 Left: OFFSETGAPTYPE=0: polyline segments are extended to close gaps Middle: OFFSETGAPTYPE=1: filleted gaps (the radius of the arc segments equals the offset distance) Right: OFFSETGAPTYPE=2: chamfered gaps (the perpendicular distance to the chamfers equals the offset distance)
Using the Quad:	
If the *Select Faces* flag of of the SELECTIONMODES system variable is set. 	Offset of the face of a solid. 
If the *Select Detected Boundaries* flag of the	Offset of a detected boundary between two overlapping squares:

SELECTIONMODES system variable is set.	 Offset of a detected boundary on the face of a solid: 

Procedures

Making a parallel copy at a specified distance
Making a parallel copy through a point
Making a parallel copy of a 3D solid edge
Making a parallel copy a 3D solid face
Making a parallel copy of a detected boundary

Related Command

Copy - makes copies of entities.

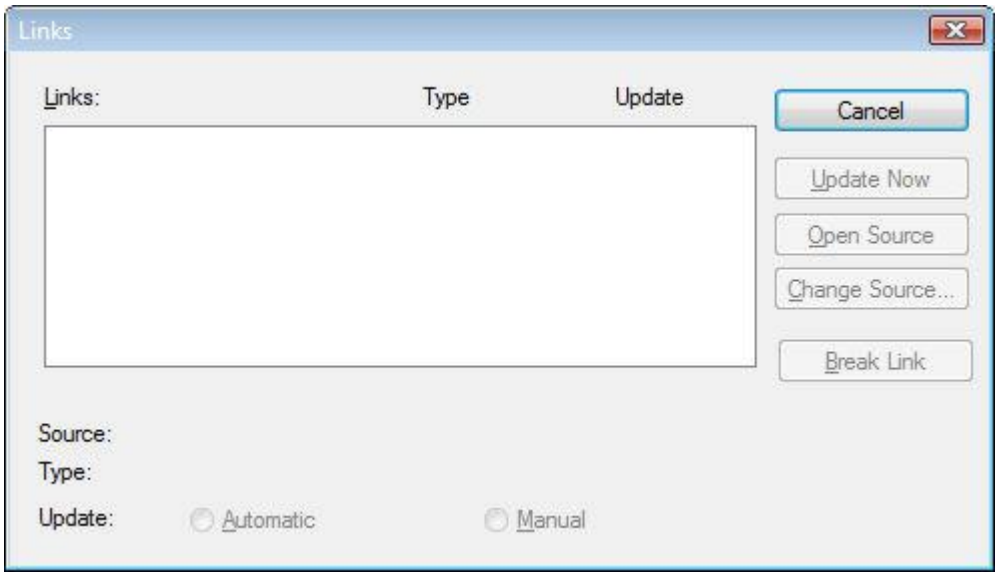
OleLinks

Adjusts links of OLE entities embedded in or linked to drawings (short for "object linking and embedding").

Accessing the Command

command bar: *olelinks*
menu bar: *Edit | Links*

: ***olelinks***
Displays a dialog box:



Change a setting for a link, and then click Update Now.

Use the **InsertObj** command to insert OLE objects in drawings.

Command Options

Option	Description
Update Now	Updates linked OLE entities. When the source file changes, use this button to update the copy in the BricsCAD drawing.
Open Source	Opens the source file in the source application. This allows you to edit the source file. When done editing, choose Update & Exit from the File menu.
Change Source	Changes the source for the linked OLE entity.
Break Link	Breaks the link with the source file. When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.
Source	Reports the path to the source file name.
Type	Reports the file type.

Update	Determines how the OLE entity in the drawing is updated, when the source changes: • Automatic - the OLE entity in the drawing changes whenever the source file changes. • Manual - the OLE entity in the drawing changes when you click the Update Now button.

Related Commands

OleOpen - opens linked and embedded objects in the appropriate application.
PasteSpec - pastes entities in several formats.
InsertObj - inserts foreign (non-BricsCAD) objects into drawings
Insert - inserts DWG files in drawings.
ImageAttach - attaches raster images to drawings.
XAttach - attaches DWG files to drawings.

OleOpen

Opens linked and embedded objects in the appropriate application.

Accessing the Command

command bar: *oleopen*

: ***oleopen***

Prompts you:

Select OLE item to open: (Select an OLE object in the drawing.)

The selected OLE object is opened.

Related Commands

OleLinks - adjusts links of OLE entities embedded in or linked to drawings
PasteSpec - pastes entities in several formats.
InsertObj - inserts foreign (non-BricsCAD) objects into drawings

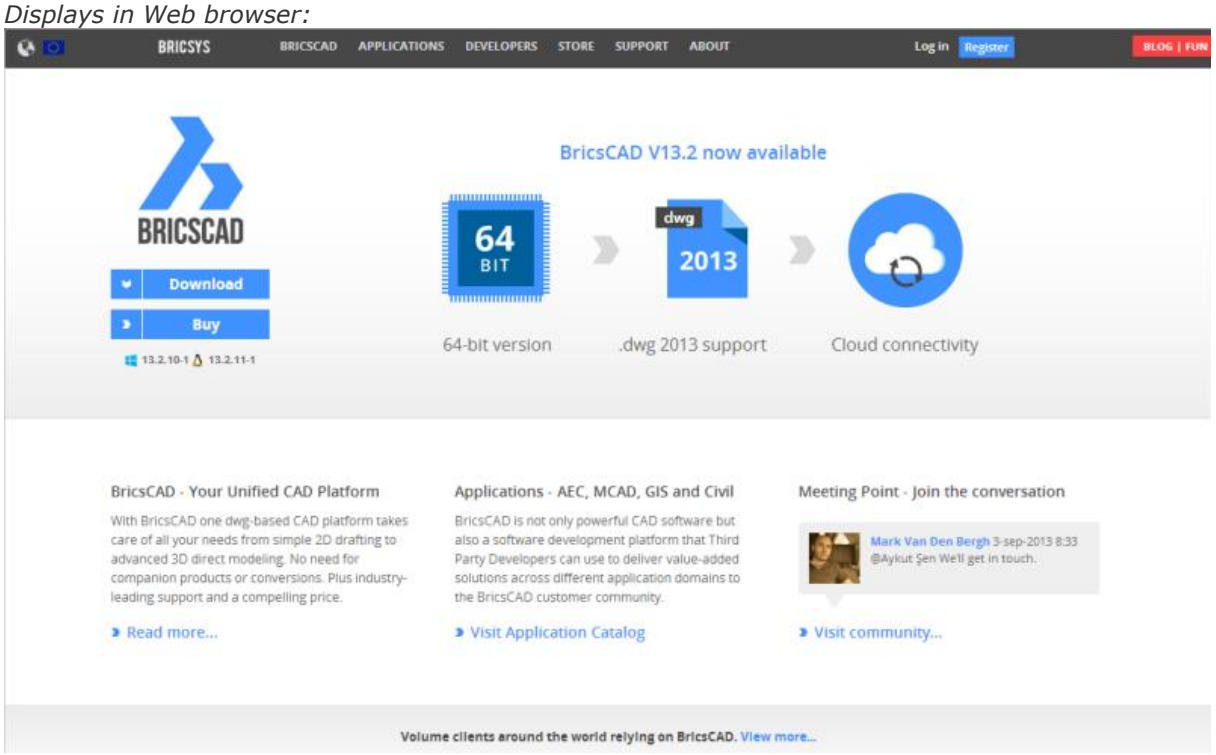
OnWeb

Opens the Bricsys home page in your computer's default Web browser.

Accessing the Command

command bar: *onweb*

: ***onweb***
Displays in Web browser:



Command Options

No options.

Related Commands

Browser - opens the default Web browser.
Url - opens the default Web browser.
Mail - attaches the current drawing to a message in the default email client.

Oops

Un-erases the last erased entity, including those erased by the Block command.

Accessing the Command

command bar: *oops*
aliases: *undelete, unerase*
menu bar: *Name*

: ***oops***

Does not prompt you in the command bar.

If the drawing contains no erased entities, BricsCAD reports, "There is nothing to erase."

Command Options

No options.

Related Commands

U - undoes the last command.
Undo - undoes one or more commands.
Redo - redoes the last undone command.
Zoom - utilizes the Previous option to show the previous view.

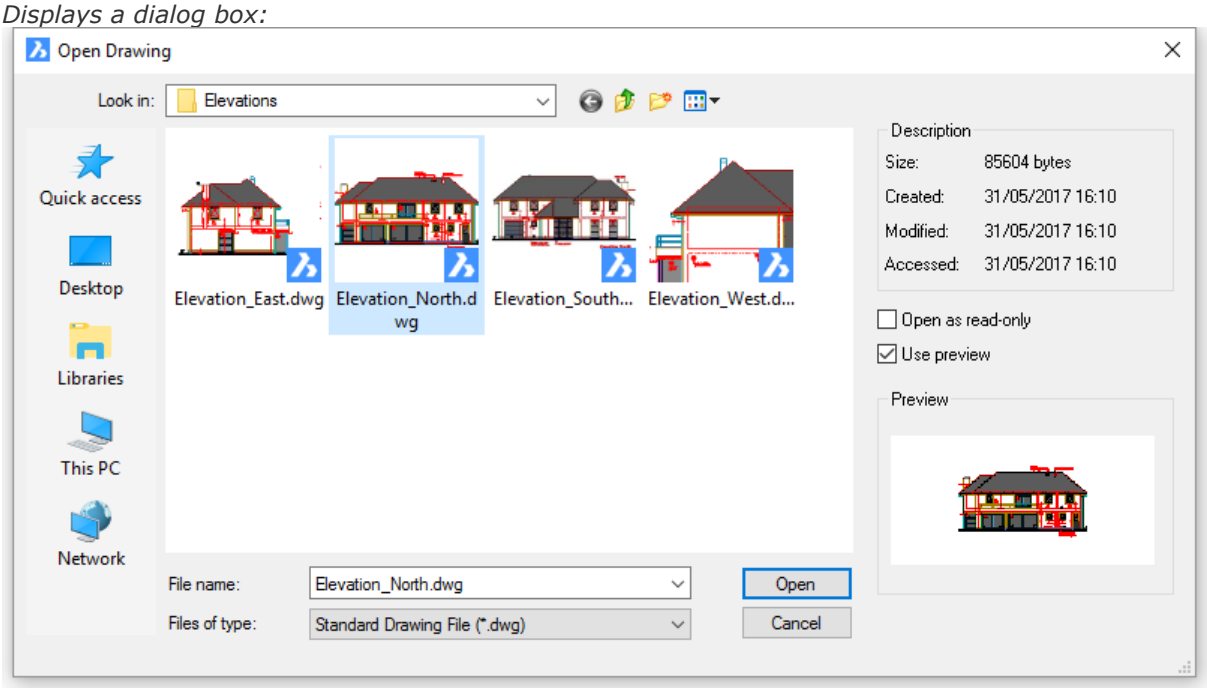
Open

Opens drawing (DWG), template (DWT), and interchange (DXF) files.

Accessing the Command

command bar: *open*
shortcut: *Ctrl+O*
aliases: *op, im, import*
menu bar: *File | Open*
toolbar: *Standard |*

: ***open***
Displays a dialog box:



Choose a file, and then click Open.

Command Options

Option	Description
Look in	Selects the drive and folder.
File name	Specifies the name of the file.
Files of type	Selects the type of file: • DWG - BricsCAD drawing file • DWT - template file • DXF - interchange file
Open	Opens the file.
Cancel	Closes the dialog box without opening a file.
Open as read-only	Opens the drawing in read-only mode. The drawing cannot be saved to the same file name.
Use Preview	Displays the preview image of the drawing.

Procedures

Working with document tabs

Related Commands

Close - closes the current drawing.
Quit - exits BricsCAD after closing open drawings.
New - starts new drawings.
SecurityOptions - sets a password to protect a drawing
WhoHas - reports the ownership of a drawing file.
WorkSets - creates and loads a set of drawing files. Allows to work with a named group of drawings.

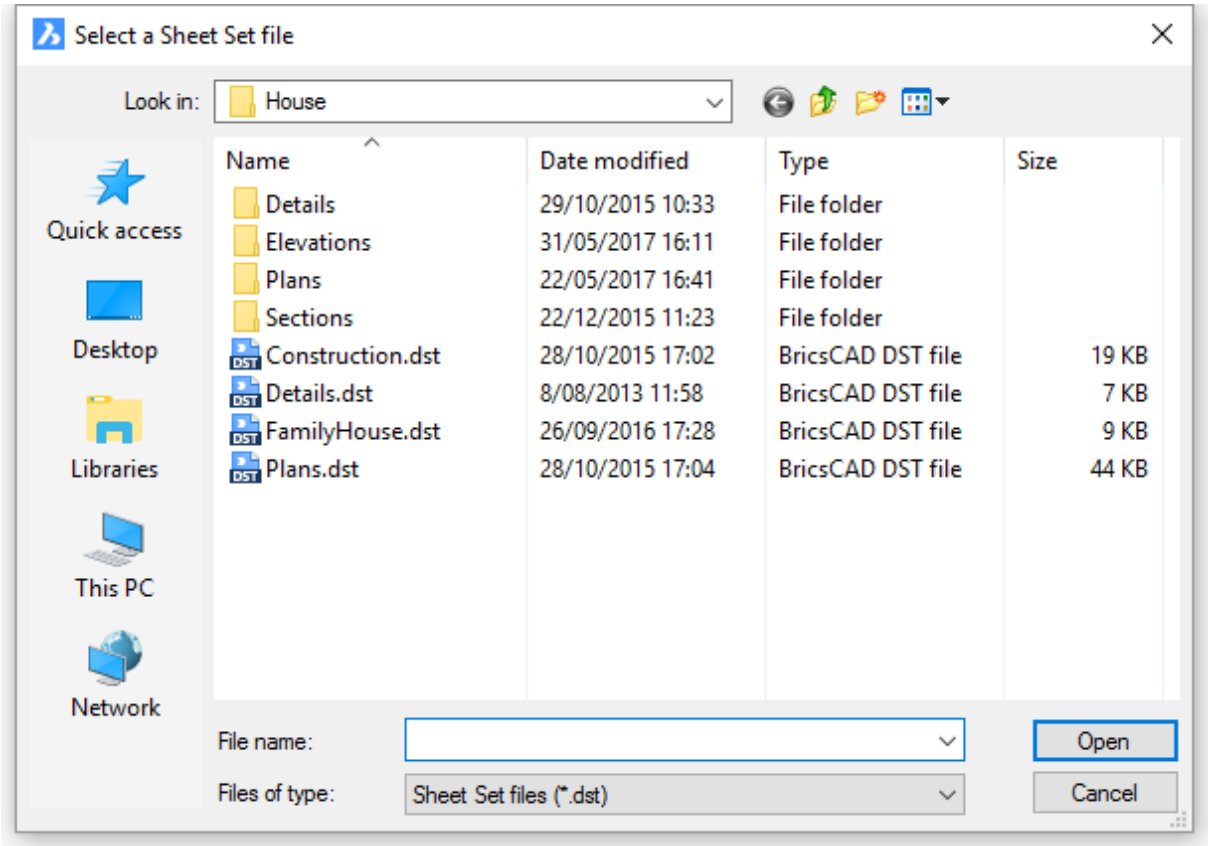
OpenSheetSet

Opens an existing sheet set; displays a file dialog box.

Accessing the Command

command bar: *opensheetset*
toolbar: *Drawing Explorer - Sheets* |

: ***opensheetset***
Displays a dialog box:



Select a sheet set file (*.dst), then double click or press the Open button.
The *Drawing Explorer - Sheets* dialog displays. See the SheetSet command.

Command Options

No options.

Related Commands

NewSheetSet - creates a new sheet set.
-OpenSheetSet - opens an existing sheet set; prompts you in the command bar.
SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.

-OpenSheetSet

Opens an existing sheet set; prompts you in the command bar.

Accessing the Command

command bar: *-opensheetset*

: ***-opensheetset***
Prompts you in the command bar:
Enter sheet set file name: (Type the path and file name of the sheetset file in the command bar.)
The *Drawing Explorer - Sheets* dialog displays. See the SheetSet command.

Command Options

No options.

Related Commands

OpenSheetSet - opens an existing sheet set; displays a file dialog box.
NewSheetSet - creates a new sheet set.
SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.

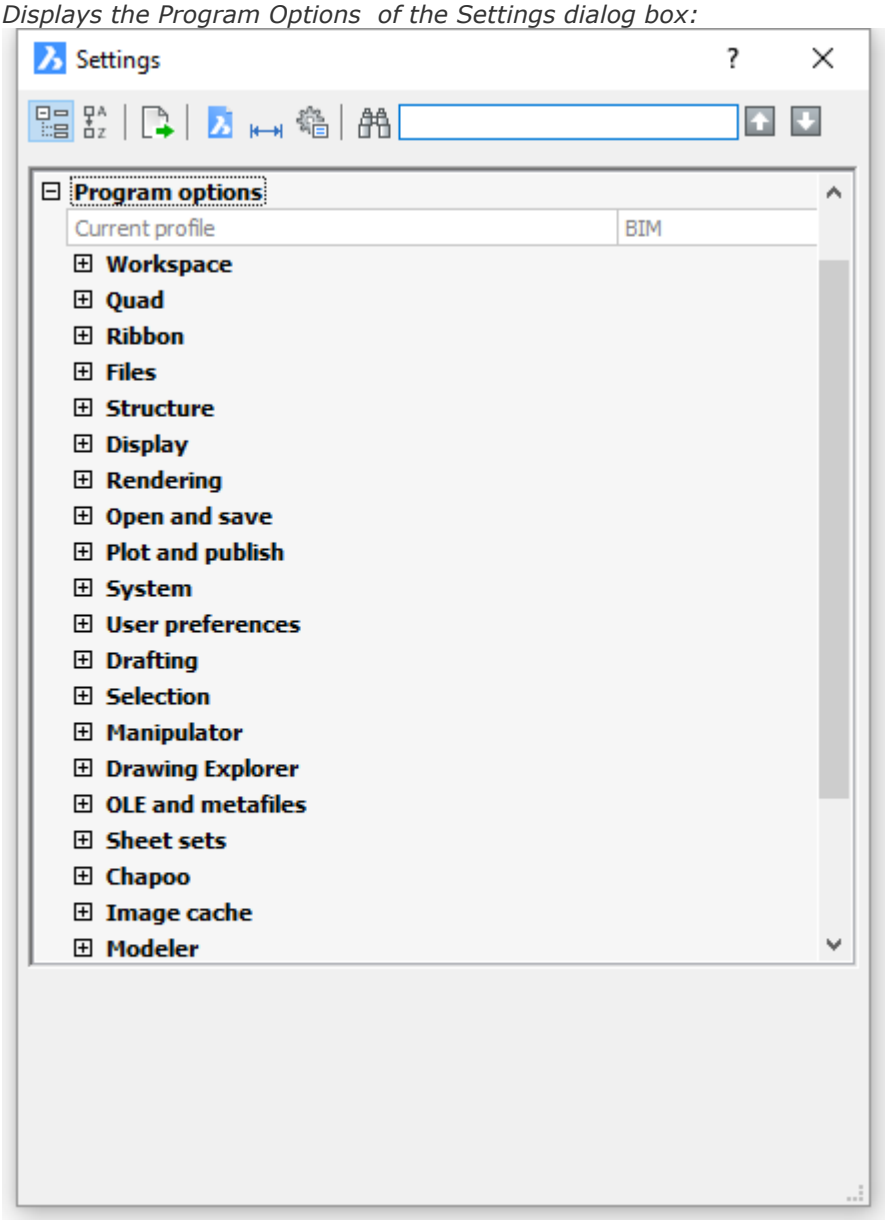
Options

Sets program options through the Settings dialog box.

Accessing the Command

command bar: *options*
aliases: *cfg, config, preferences, prefs,* ***settings***

: ***options***
Displays the Program Options of the Settings dialog box:



Change a setting, and then click ***X****.*

Command Options

Option	Description
Current profile	Specifies the name of the profile file, which stores all setting applicable to the current user.

Related Commands

Settings - changes the values of system variables and user preferences through a dialog box.

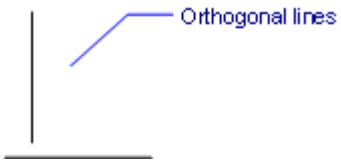
Orthogonal

Constrains drawing and editing to 90-degree angles.

Accessing the Command

command bar: *orthogonal*
transparent: *'orthogonal*
shortcut: *F8*
aliases: *or, ortho*
menu bar: *Settings | Draw Orthogonal*
toolbar: *Settings |*
status bar: *ORTHO*

: ***orthogonal***
Prompts you in the command bar:
ORTHOMODE is currently off: ON/Toggle/<Off>: ***(Enter an option.)***

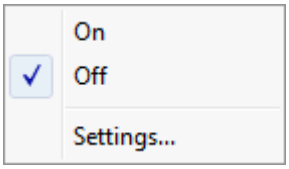


Command Options

Option	Description
ON	Turns on orthogonal mode.
Toggle	Toggles orthogonal mode: if on, turns it off; if off, turns it on.
OFF	Turns off orthogonal mode.

Status Bar Menu

Right-click ORTHO on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on orthographic mode.
Off	Turns off orthographic mode.
Settings	Opens the Coordinate Input section of the Settings dialog box. ⊟ **Drawing** ⊟ **Drafting** ⊞ **Drawing units** ⊟ **Coordinate input** Orthogonal mode ☐ Orthogonal mode
Orthogonal mode	Toggles orthogonal mode: • On - turns on ortho mode. • Off - turns off ortho mode.

Related Commands

Snap - sets the ortho angle.
Settings - sets the parameters for ortho mode.

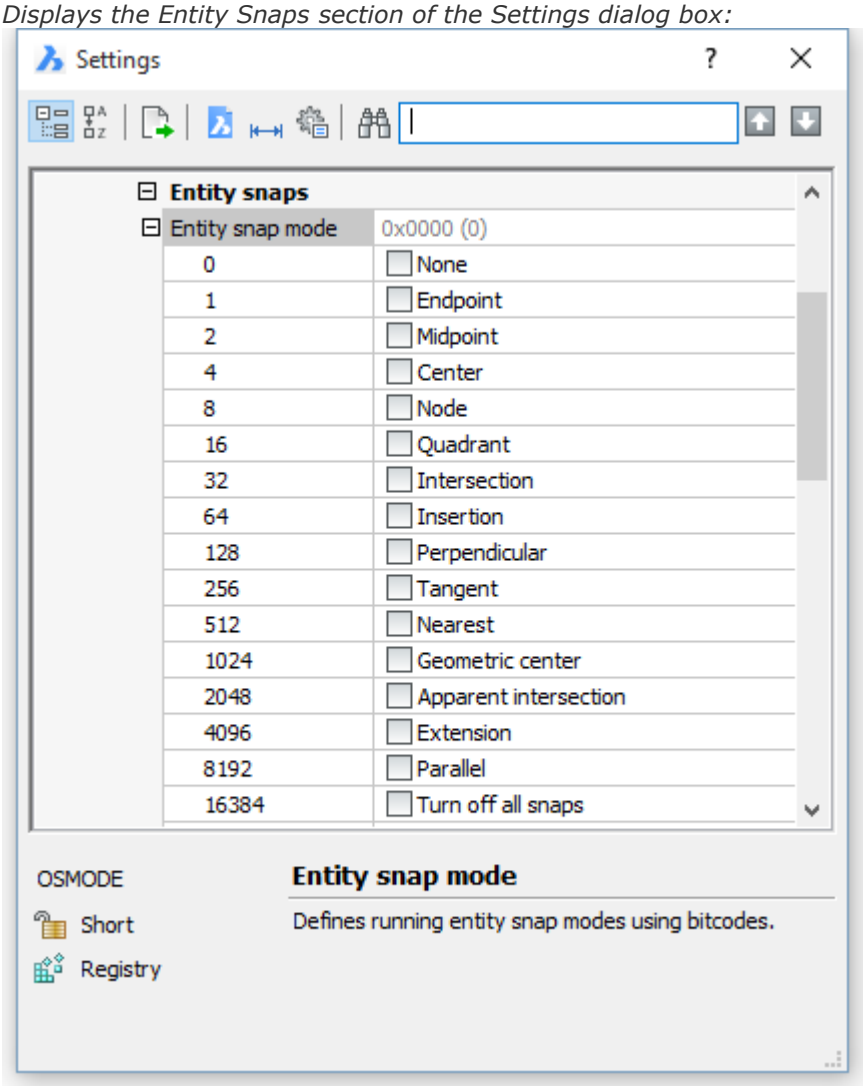
OSnap

Sets entity snaps through the Settings dialog box (short for "object snap").

Accessing the Command

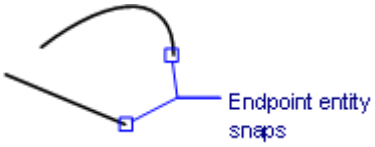
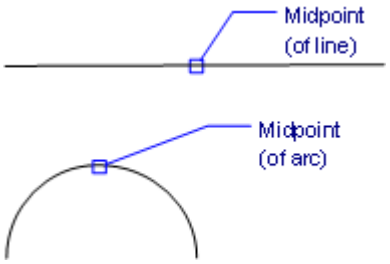
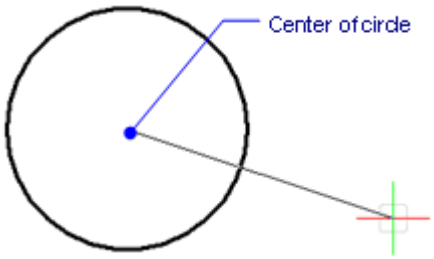
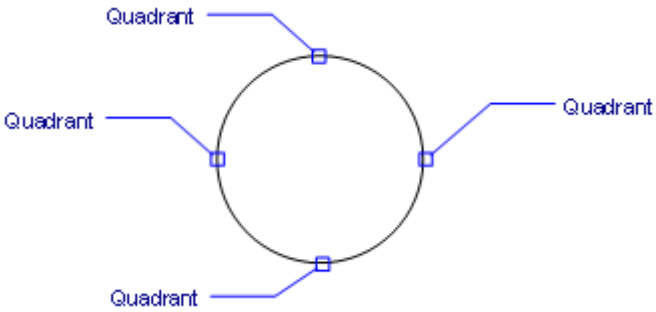
command bar: *osnap*
transparent: *'osnap*
shortcut: *F3*
alias: *ddosnap, ddesnap, os, setesnap*
menu bar: *Settings | Entity Snap Settings*
toolbar: *Settings |*
status bar: *ESNAP*

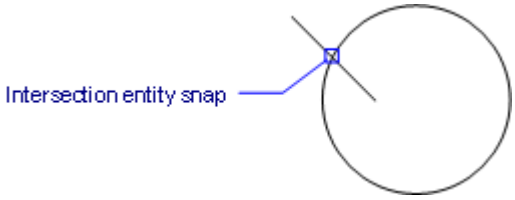
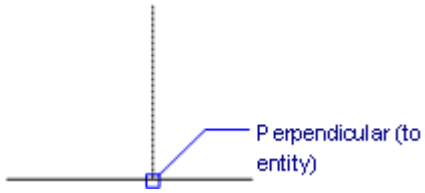
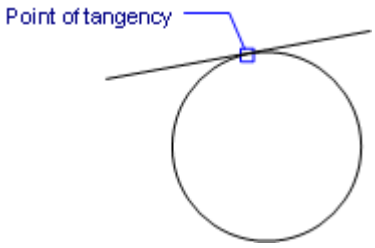
: ***osnap***
Displays the Entity Snaps section of the Settings dialog box:

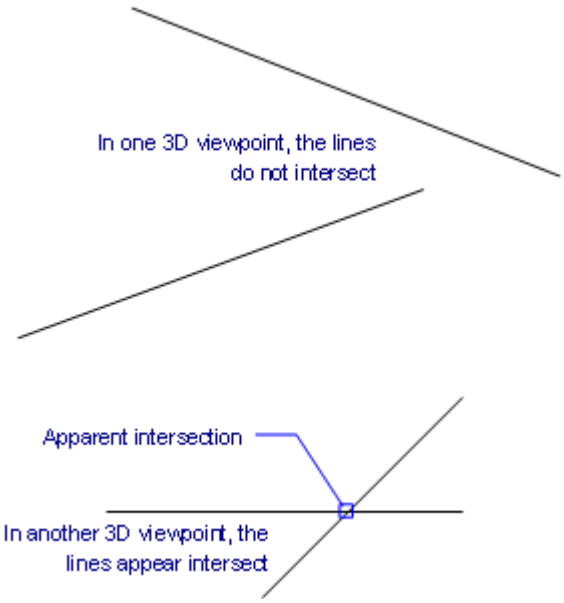
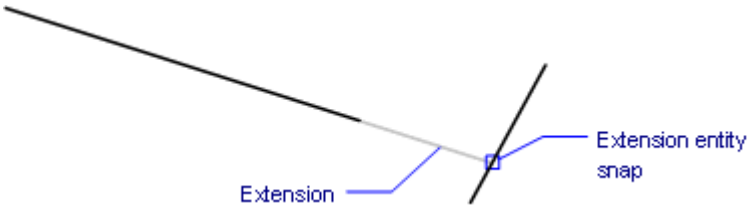
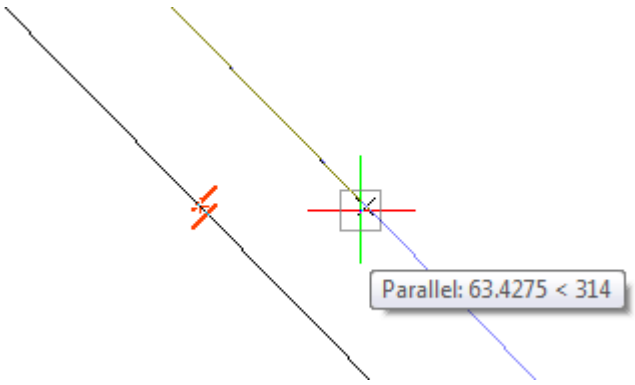


*Change a setting, and then click **X**.*

Command Options

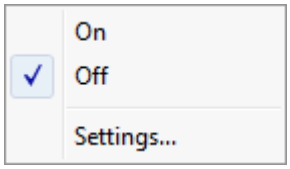
Option	Description
1 - Endpoint	Toggles endpoint entity snap mode. Snaps to the end points of open entities, such as lines,arcs, open polylines,and open splines.  Can also be toggled with the **Endpoint** command.
2 - Midpoint	Toggles midpoint entity snap mode. Snaps to the midpoint of open entities.  Can also be toggled with the **Midpoint** command.
4 - Center	Toggles center entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs.  Can also be toggled with the **Center** command.
8 - Node (Point)	Toggles node entity snap mode. Snaps to point entities. Can also be toggled with the **Node** command.
16 - Quadrant	Toggles quadrant entity snap mode. Snaps to the quadrants points of circular entities, such as circles, arcs, and polyarcs.  Can also be toggled with the **Quadrant** command.

32 - Intersection	Toggles intersection entity snap mode. Snaps to the intersection of any pair of entities.  Can also be toggled with the **Intersection** command.
64 - Insertion	Toggles insertion entity snap mode. Snaps to the insertion point of blocks and text. Can also be toggled with the **Insertion** command.
128 - Perpendicular	Toggles perpendicular entity snap mode. Snaps to the perpendicular of one entity to another.  Can also be toggled with the **Perpendicular** command.
256 - Tangent	Toggles Tangent entity snap mode. Snaps to the tangencies of circular entities.  Can also be toggled with the **Tangent** command.
512 - Nearest	Toggles nearest entity snap mode. Snaps to the nearest point on any entity. Can also be toggled with the **Nearest** command.
1024 - Geometric Center	Toggles geometric center entity (GCenter) snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. Can also be toggled with the GCenter command.
2048 - Apparent Intersection	Toggles Apparent entity snap mode. Snaps to the apparent intersection of entities in 3D space.

	 Can also be toggled with the **Apparent** command.
4096 - Extension	Toggles extension entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection.  Can also be toggled with the **Extension** command.
8192 - Parallel	Toggles parallel entity snap mode. Snaps to the parallel point of entities. Allows to draw lines, polyline segments, xlines and rays parallel to another entity.  Can aso be toggled with the **Parallel** command.
16384 - Toggle all snaps	Turns all entity snap modes off. Modes can also be turned off with the **None** command.

Status Bar Menu

Right-click ESNAP on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on entity snaps.
Off	Turns off entity snaps.
Settings	Opens the Entity Snaps section of the Settings dialog box.

Related Commands

-OSnap - toggles entity snap modes in the command bar.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

-OSnap

Toggles entity snap modes at the command line (short for "object snap").

Accessing the Command

command bar: *-osnap*
transparent: *'-osnap*
shortcut: *F3*
aliases: *esnap, -os*
status bar: *ESNAP*

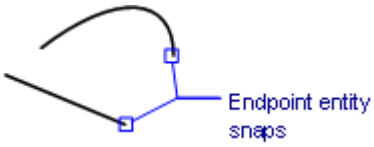
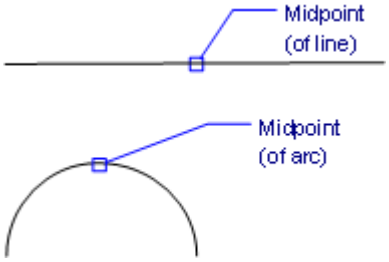
: ***-osnap***
Prompts you in the command bar:

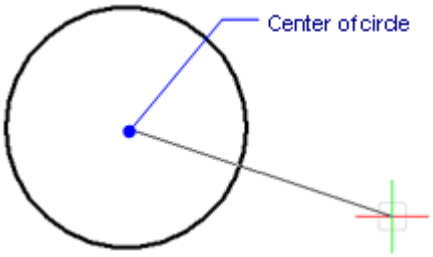
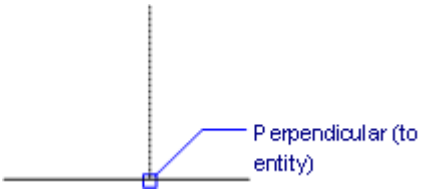
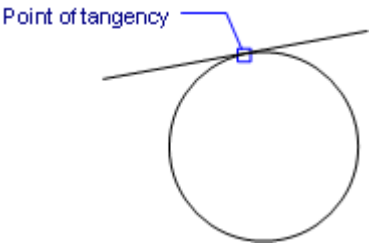
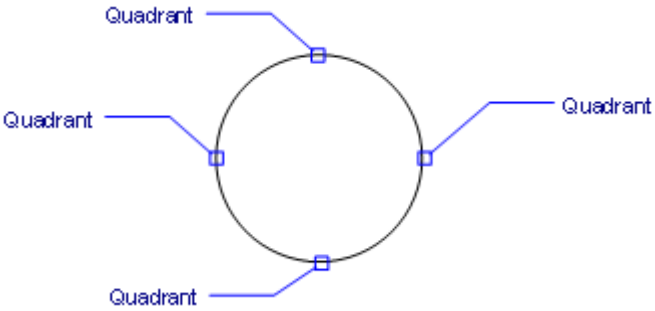
Snaps:
NEArest/ENDpoint/MIDpoint/CENter/GeometricCEnter/PERpendicular/TANgent/QUAdrant/INSertion/POInt/INTersection/APParent/ON/<OFF>: *(Enter an option, or press Enter to toggle entity snaps on or off.)*

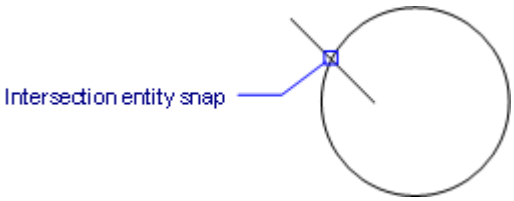
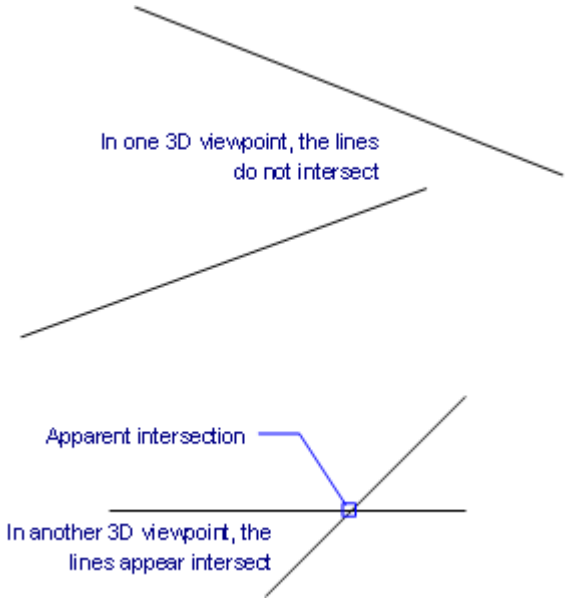
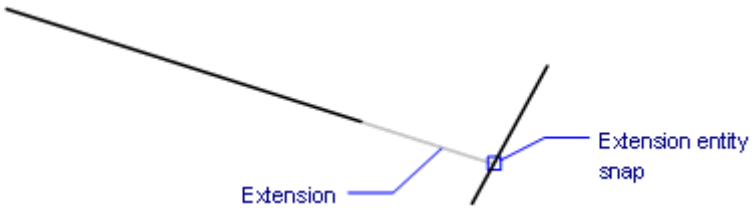
The following entity snaps work with 3D solid models:

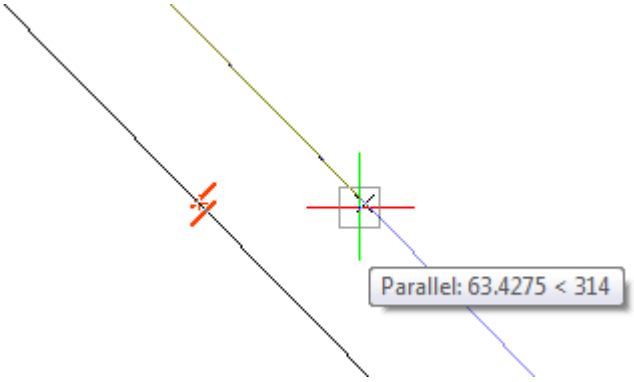
- Endpoint
- Midpoint
- Nearest

Command Options

Option	Description
NEArest	Toggles nearest entity snap mode. Snaps to the nearest point on any entity. Can also be toggled 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.  Can also be toggled with the Endpoint command.
MIDpoint	Toggles midpoint entity snap mode. Snaps to the midpoint of open entities.  Can also be toggled with the Midpoint command.

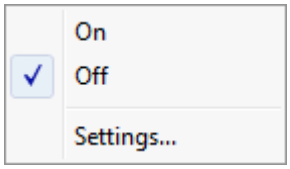
CENter	Toggles center entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs.  Can also be toggled with the Center command.
GeometricCEnter	Toggles geometric center entity (GCenter) snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. Can also be toggled with the GCenter command.
PERpendicular	Toggles perpendicular entity snap mode. Snaps to the perpendicular of one entity to another.  Can also be toggled with the Perpendicular command.
TANgent	Toggles Tangent entity snap mode. Snaps to the tangencies of circular entities.  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.  Can also be toggled with the Quadrant command.

INSertion	Toggles insertion entity snap mode. Snaps to the insertion point of blocks and text. Can also be toggled with the Insertion command.
POInt (Node)	Toggles node entity snap mode. Snaps to point entities. Can also be toggled with the Node command.
INTersection	Toggles intersection entity snap mode. Snaps to the intersection of any pair of entities.  Can also be toggled with the Intersection and 3dIntersection commands.
APParent	Toggles Apparent entity snap mode. Snaps to the apparent intersection of entities in 3D space.  Can also be toggled with the Apparent and 2dIntersection commands.
EXTension	Toggles extension entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection. 

	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.  Can aso be toggled with the **Parallel** command.
ON	Turns all entity snap modes on.
OFF	Turns all entity snap modes off. Modes can also be turned off with the None command.

Status Bar Menu

Right-click ESNAP on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on entity snaps.
Off	Turns off entity snaps.
Settings	Opens the Entity Snaps section of the Settings dialog box.

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

Overkill

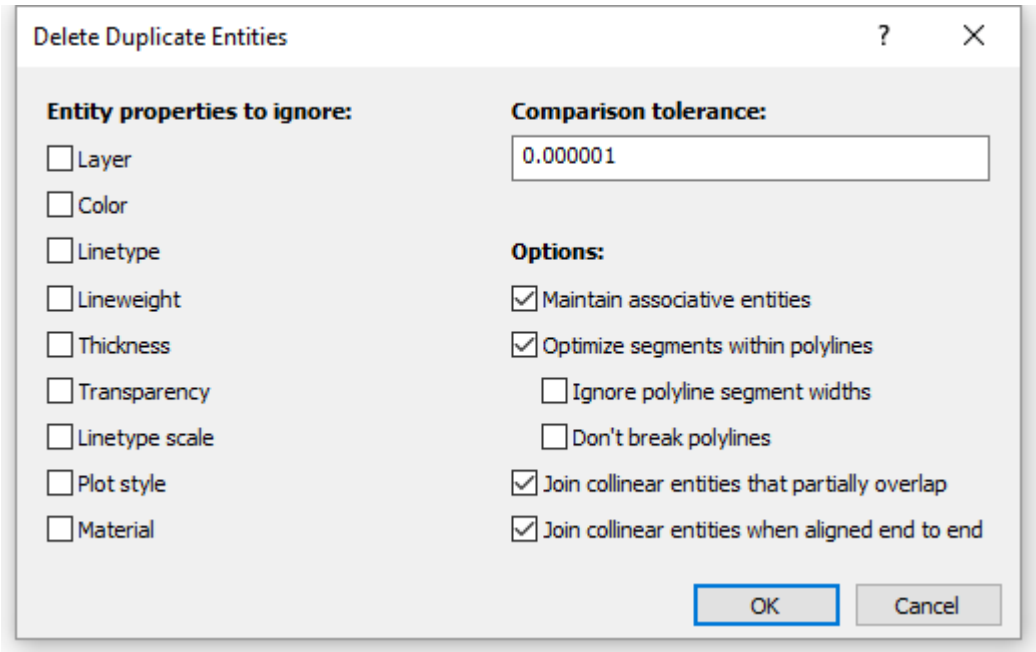
Deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones.

Accessing the Command

command bar: *overkill*
menu: *Edit | Delete Duplicate Entities*

: ***overkill***
Prompts you in the command bar:
Select entities: *(Compose a selection set, then press Enter.)*

Displays a dialog box:



Command Options

Option	Description
Entity properties to ignore	Selected properties are ignored when comparing duplicate or overlapping entities.
Comparison tolerance	Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.
Options	If checked: • **Maintain associative entities**: associative entities are not processed. • **Optimize segments within polylines**: segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. If lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline, unless the *Don't break polylines* option is checked.

	• **Ignore polyline segment widths**: ignores the *Width* property of polyline segments. • **Don't break polylines**: polyline entities are kept intact. • **Join collinear entities that partially overlap**: partially overlapping entities are unified into a single entity. • **Join collinear entities when aligned end to end**: collinear lines and polylines having coinciding endpoints are unified into a single entity or segment.

Related Commands

-Overkill - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones; prompts you in the command bar.

Break - erases a portion of an entity.
Erase - erases entities from the drawing.
Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
Oops - returns erased entities.
Purge - removes unused named entities from drawings, such as unused layers and linetypes.
Trim - removes portions of entities.
U - unerases erased entities.
Undo - unerases erased entities.

-Overkill

Deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones.

Accessing the Command

command bar: *-overkill*

: ***-overkill***
Prompts you in the command bar:

Select entities: (Compose a selection set, then press Enter.)

Enter an option to change [Done/Ignore/tOlerance/optimize Plines/combine parTial overlap/combine Endtoend/Associativity] <done>: (Enter an option or press Enter to accept the default settings.)

Command Options

Option	Description
Done	Executes the overkill procedure.
Ignore	Sets the entity properties to ignore. Prompts you: [None/All/Color/LAyer/Ltype/ltScale/LWeight/Thickness/TRansparency/plotSTyle/Material] <None>: **Enter an option.**
Tolerance	Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process. Prompts you:
Optimize Plines	Segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. If lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline, unless the *Don't break polylines* option is checked. Prompts you: Optimize segments within plines [segment wiDth/Break polyline/Yes/No] <Yes>: Enter an option. • Segment width: Prompts you: Ignore segment widths [Yes/No] <No>: If *Yes*, ignores the *Width* property of polyline segments. • Break polylne: Prompts you: Break polylines [Yes/No] <Yes>: If *No*, polyline entities are kept intact • Yes: Optimizes polyline entities. • No: Polyline entities are not optimized..
Combine partial overlap	Partially overlapping entities are unified into a single entity. Prompts you: Combine co-linear entities that partially overlap [Yes/No] <Yes>:
Combine End to End	Collinear lines and polyline segments having coinciding endpoints are unified into a single entity or segment. Prompts you: Combine co-linear entities when aligned end to end [Yes/No] <Yes>:

Associativity	Associative entities are not processed. Prompts you: Maintain associative entities [Yes/No] <Yes>:

Related Commands

Overkill - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones; displays a dialog box.

Break - erases a portion of an entity.
Erase - erases entities from the drawing.
Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
Oops - returns erased entities.
Purge - removes unused named entities from drawings, such as unused layers and linetypes.
Trim - removes portions of entities.
U - unerases erased entities.
Undo - unerases erased entities.

P

PageSetup

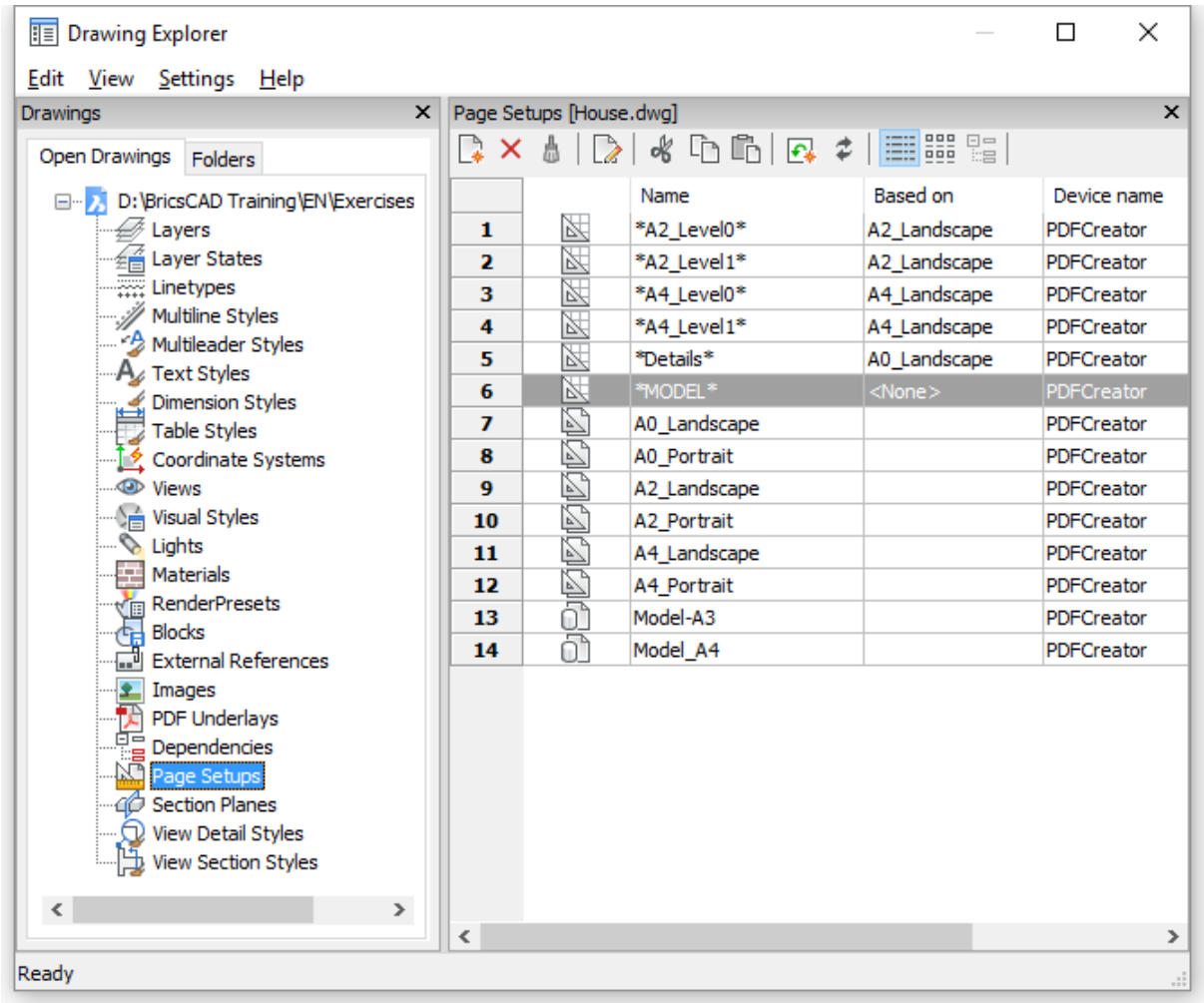
Creates and edits page setups for plotting drawings in the Drawing Explorer.

Accessing the Command

command bar: *pagesetup*
menu bar: *File | Page Setup*
toolbar: *Layout |*

: ***pagesetup***

Displays the Drawing Explorer - dialog box:

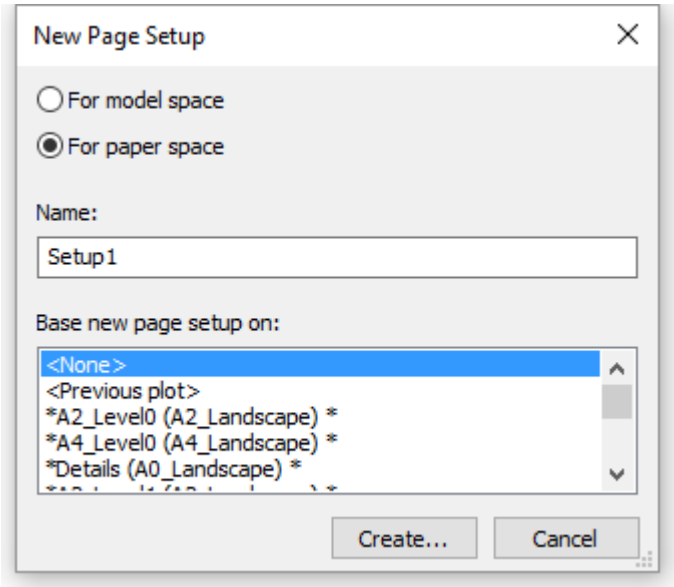


Do one of the following:

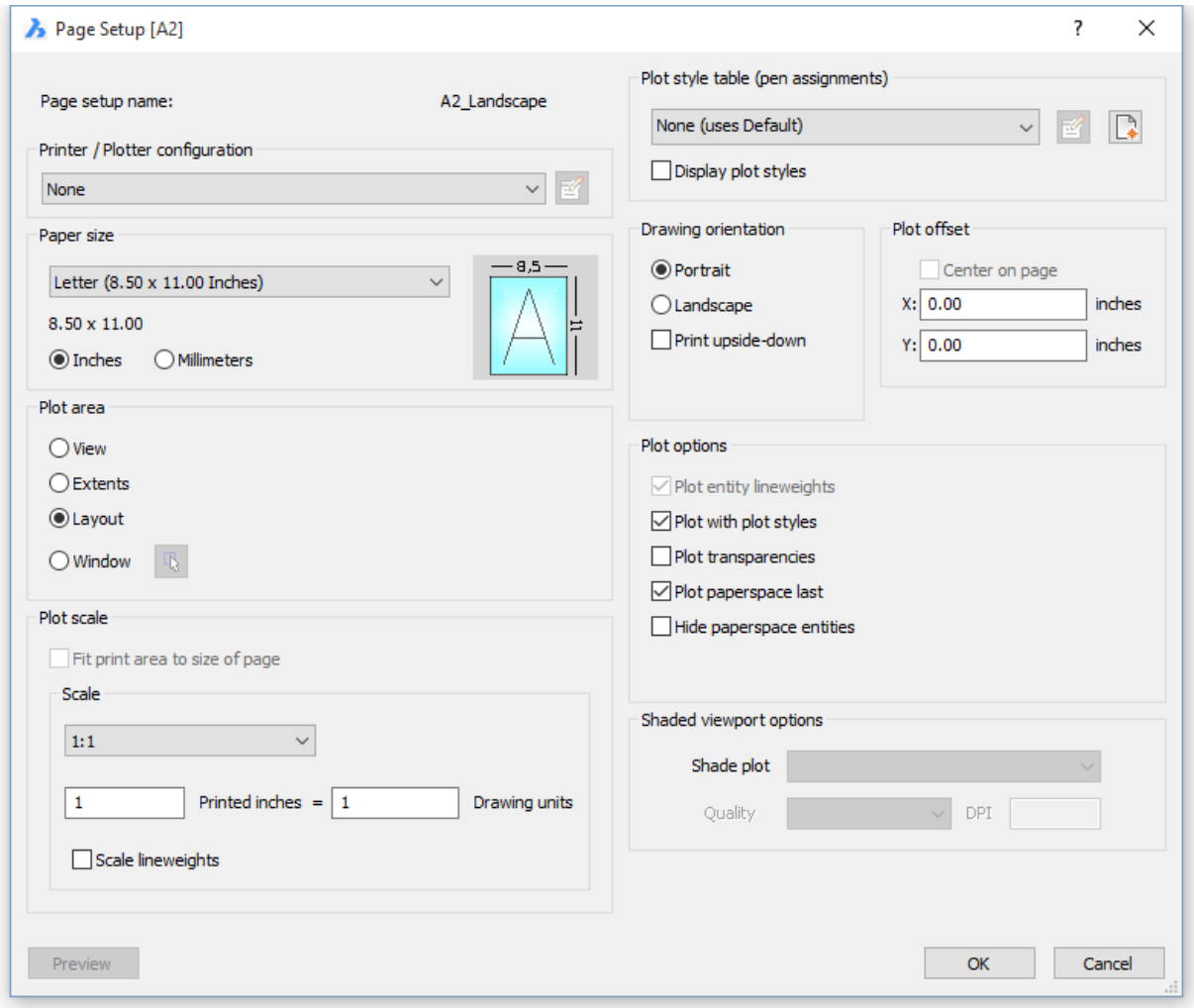
- Click the *New* icon ( ) to create a new page setup.
- Double click the icon ( ) of a **Layout** or **Model** to edit the page setup of a layout or the model space.
- Select one or more layouts ( ) then click the *Delete* button ( ) to remove layouts.
- Double click the icon ( ) of a model space page setup to edit the definition.

- Double click the icon ( ) of a paper space page setup to edit the definition.

When creating a new page setup you have to choose between *Model space* or *Paper Space*, then click the Create... button.



The Page Setup dialog box displays:



*Choose options, and then click **OK**.*

Command Options

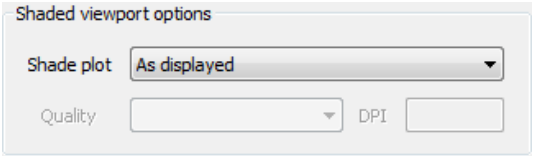
Option	Description
Print / Plotter Configuration	
Printer / Plotter configuration \\Ralph\hp LaserJet 1320 PCL 6 None (uses Default) Microsoft XPS Document Writer Adobe PDF \\Ralph\hp LaserJet 1320 PCL 6 NewPlotterCfg.pc3	Selects the printer or plotter. BricsCAD works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices. It also uses predefined printer parameters stored in PC3 files. See **PlotterManager** command.
Edit plotter configuration	Opens the printer's Properties dialog box for customizing the printer's parameters.
Paper Size	

Paper size Letter 8K 273x394 mm 16K 197x273 mm User Defined 8.5x13 (Custom) B5 (ISO) B6 (ISO) Double Japan Postcard Rotated A6 Japanese Postcard Envelope Monarch Envelope B5 Envelope C5 Envelope DL	Chooses the size of paper. You can select standard sizes listed by the droplist; these are sizes supported by the printer. While printer may appear to support many different sizes, you should select only the size of paper that is actually in the printer. If User Defined appears in the list, the paper's dimensions are specified by the printer's Properties dialog box. The image indicates the size, position and orientation of the current plot area on the selected paper size.
Units 8.50 x 11.00 Inches Millimeters	Switches paper measurements between imperial and metric. 215.90 x 279.40 Inches Millimeters
Plot Area	
Plot area View Extents Limits Window	Specifies which area of the drawing to print: • View - prints the current view or a named view. See View command. Choose a view from the droplist: View Current view Current view Detail Front View • Extents - prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not taken into account when calculating the extents. • Limits - prints the limits of the drawing, as specified by the Limits command. • Window - prints a rectangular area of the drawing. You define the rectangular area by entering x,y coordinates or by clicking the Select Area to be Printed button.
Select area to be printed	Specifies the rectangular area to be printed. Click the button; the dialog box is dismissed temporarily, and you are prompted: Select first corner of window - specify one corner of the rectangle. Select second corner of window - specify the other corner. You can pick points or enter x,y coordinates. The Print dialog box returns, with the pair of x,y coordinates added to the Windows Plot Area boxes.

Windowed plot area X: 4.777 Y: 10.949 X: 9.735 Y: 15.907	Specifies the x,y coordinates of the lower-left and upper-right corners of the rectangular area to be printed. You can enter x,y coordinates, or click the button to pick points in the drawing.
Plot Scale	
Plot scale ☑ Fit print area to size of page	Scales the drawing so that it fits the printable area of the paper. The scale is calculated automatically by BricsCAD, and takes into account the print area of the drawing. The printable area is the size of the paper, minus the margins. The margins are the strips along the four edges that the printer uses for handling the paper. When this option is turned on, you cannot specify the scale factor.
Scale 1:1 Custom 1:1 1:2 1:4 1:8 1:10	Specifies the scale factor of the printed drawing. Choose a scale factor from the droplist, or select *Custom* to specify a scale factor. Available scales in the list can be edited using the ScaleListEdit command. For scale factors like 1:5, the drawing is scaled smaller. For scale factors like 5:1, the drawing is scaled larger.
1 Printed inches = 1 Drawing units	In Model Space page setups: Specifies user-defined scale factors. For example, to print a drawing of 50'-long house on paper 11" wide, enter: Printed Inches = 11" Drawing Units = 50' BricsCAD works out the scale factor automatically. In Paper Space page setups: The Scale setting specifies the relation between paper units and drawing units. Paper units are either inches or millimeters. Drawing units can be feet, centimeters, meters, ... Scale setting when drawing units are feet: 12 Printed inches = 1 Drawing units Scale setting when drawing units are cm: 10 Printed mm = 1 Drawing units
Scale lineweights	If on, lineweights are scaled with respect to the plot scale.
Plot Style Table	

Plot style table (pen assignments) None (uses Default) None (uses Default) Bricscad.ctb Default.ctb MonoChrome.ctb	Specifies the plot style table to use, which assigns properties to "pens," colors, and entities. If the drawing does not use plot styles, then only CTB (color-based table) files are listed. If the drawing uses plot styles, then only STB (style-based table) files are listed. Plot style table (pen assignments) Default.stb None (uses Default) Bricscad.stb Default.stb MonoChrome.stb When you switch from Non to a named plot style, BricsCAD prompts you: Assign plot style table to all layouts? - click Yes or No. • Yes - assigns the CTB or STB plot style file to all layouts. • No - assigns the plot style file to the current layout only.
Edit Plot Style	Opens the Plot Style Table editor dialog box. See PlotStyle command. This button is available only when the drawing uses STB plot styles.
Create New Plot Style	Creates new plot styles; runs the Add Plot Style Table wizard. If 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). See **StylesManager** command.
Display plot styles	Toggles the display of plot style names.
Drawing Orientation	
Drawing orientation Portrait Landscape Print upside-down	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.
Plot Offset	
Plot offset X: 0 mm Y: 0 mm	Specifies the offset distance for the print. Enter positive or negative distances to move the drawing in the x and/or y distances:

	• Positive values - 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.
Center on page	Centers the print on the page. The center of the drawing is positioned in the center of the paper. Together with Fit Print Area to Size of Page, these options are excellent for prints were scale is unimportant.
Plot Options	
Plot entity lineweights	Toggles the use the lineweights: • On - mimics lineweights in the print. • Off - ignores lineweights. This option is unavailable when Plot with Plot Styles is turned OFF.
Plot with plot styles	Toggles the use of plot styles: • On - uses plot styles determine the look of the printed drawing. • Off - does not use plot styles. Plot styles override lineweight settings.
Plot transparencies	Plots entities using the value of the Transparency property.
Plot paperspace last	Toggles printing order: • On - prints model space entities first, followed by paper space entities. • Off - prints paper space entities first, followed by entities in model space. This option is available only when printing layouts; it is unavailable in model tab.
Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.

Shaded Viewport Options	
	Allows to override the visual style of the current view when printing model space. This option is disabled for paper space page setups. 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).
Preview	Displays a preview of the print. In the preview window, click the Print Settings button to return to this dialog box. See **Preview** command.
OK	Saves the changes and returns to the drawing.
Cancel	Returns to the drawing editor.

Related Commands

CPageSetup - edits the page setup of the current layout or model space in a dialog box.

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.
ExportPdf - exports layouts in a drawing to a pdf docucment.
PsetupIn - Imports page setup definitions from another drawing through a dialog box.
-PsetupIn - imports page setup definitions from another drawing through the command bar.

PlotStyle - sets the current plot style.
Print - plots drawings with plot styles.
Properties - changes plot styles assigned to entities.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

-Publish - prints a saved sheet list (= a list of model space or paper space layouts).
ScaleListEdit - edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport.

Explorer - Opens the *Drawing Explorer* dialog box.

Parallel

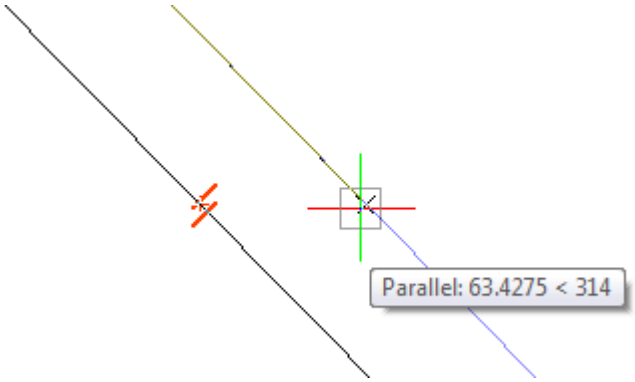
Toggles parallel entity snap; allows to draw a line, polyline segment, xline or ray parallel to another entity.

Accessing the Command

command bar: *parallel*
transparent: *par*
toolbar: *Entity Snaps* |

: ***parallel***

Displays no prompts in the command bar.



When drawing a line, polyline segment, xline or ray, hover the cursor over the entity you want to draw parallel to: the parallel snap marker displays on the entity, wait until a small cross indicates the entity is identified; a tracking line then displays when the entity you are drawing is parallel to the selected entity.

Command Options

Option	Description
Parallel	Entering this command toggles Parallel entity snap. It allows to draw parallel to another entity. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Perpendicular - toggles perpendicular snap mode
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.

Pan

Moves the entire drawing within the current viewport.

Accessing the Command

command bar: *pan*
transparent: *'pan*
alias: *p*
menu bar: *View | Zoom| Pan*
toolbar: *View | Zoom |*

: ***pan***
See RtPan command.

Related Commands

-Pan - moves the drawing within the current viewport.
RtPan - pan the drawing in realtime.
RtRot - rotates 3D drawings in real-time (short for "real time rotation").
RtRotf - rotates 3D drawings in real-time (short for "real time rotation").
RtZoom - performs real-time zooming of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.

-Pan

Moves the entire drawing within the current viewport.

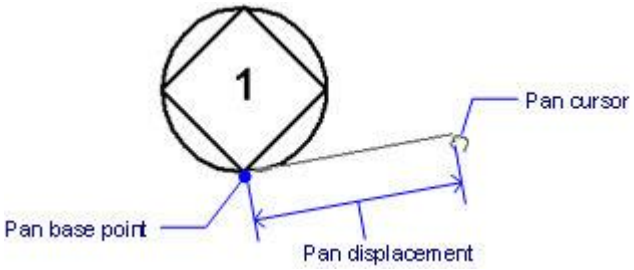
Accessing the Command

command bar: *-pan*
transparent: *'-pan*
shortcut: *PgUp, PgDn, Shift+Right, Shift+Left, Shift+Up, Shift+Down*
alias: *-p*
: ***-pan***

Prompts you in the command bar:

Left/Right/Up/Down/PGLeft/PGRight/PGUp/PGDown/<Pan base point>: **(Pick a point, or enter an option.)**

Pan displacement point: **(Pick another point.)**



Command Options

Option	Description
Pan base point	Specifies the pan distance's starting point.
Pan displacement point	Specifies the ending point.
Left	Pans the drawing 5% to the right. The drawing is panned by 5% of the width of the current viewport.
Right	Pans the drawing 5% to the left.
Up	Pans the drawing 5% downwards.
Down	Pans the drawing 5% upwards.
PGLeft	Pans the drawing 100% to the right. You can also press the Shift+Left cursor key. The drawing is panned by the entire width of the current viewport.
PGRight	Pans the drawing 100% to the left. You can also press the Shift+Right cursor key.
PGUp	Pans the drawing 100% upwards. You can also press the Shift+Up cursor key.
PGDown	Pans the drawing 100% downwards. You can also press the Shift+Down cursor key.

Related Commands

Pan moves the drawing within the current viewport.
RtPan - pan the drawing in realtime.
Zoom - increases and decreases the size of the drawing within the current viewport.

RtZoom - performs real-time zooming of the drawing.

-Parameters

Manages dimensional constraint parameters in the current drawing.

Accessing the Command

command bar: *-parameters*

: ***-parameters***
Prompts you in the command bar:

Enter parameter option: New/Edit/Rename/Delete/<?>: (Enter an option or press Enter to display a list of the currently defined parameters)

Command Options

Option	Description
New	Creates a user variable. Prompts you: Enter name for new user parameter - type a name for the new user variable. Enter expression - enter a valid mathematical expression or a fixed value.
Edit	Edits the expression for the dimensional constraint or user variable. Prompts you: Enter parameter name - type the name of a dimensional constraint parameter or a user variable. Old Expression = (d1*2 + p)/3, Value = 4.33333 Enter expression - type a new mathematical expression or press Enter to confirm the current expression.
Rename	Renames a dimensional constraint or user variable. Prompts you: Enter old parameter name - type the name of an existing dimensional constraint or user variable. Enter new parameter name - type a new name for the dimensional constraint or user variable.
Delete	Deletes a dimensional constraint or user variable. Prompts you: Enter parameter name to delete - type the name of an existing dimensional constraint or user variable.
?	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.

Related Commands

CleanUnusedVariables - purges variables not used in constraint expressions and not linked to dimensions.
-BmParameters - allows to list and edit the parameters of inserted components.
-DesignTableEdit - allows to edit designtables.

DcAligned - constrains the distance between two points.
DcAngular - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert - converts an associative dimension to a dimensional constraint.
DcDiameter - constrains the diameter of a circle or an arc.
DcHorizontal - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcLinear - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcRadius - constrains the radius of a circle or an arc.
DcVertical - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

ConstraintBar - controls the display of the constraint bar of entities.
Delconstraint - removes all dimensional and geometrical constraints from an entity.

Dimconstraint - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.
Geomconstraint - applies geometric relationships between entities and/or valid constraint points on entities.

PasteBlock

Pastes entities from the Clipboard as a block.

Accessing the Command

command bar: *pasteblock*
shortcut: *Ctrl+Shift+V*
menu: *Edit | Paste as Block*

: ***pasteblock***

If the Clipboard contains entities copied with the CopyBase command, then the entities are pasted at their base point; you are not prompted in the command bar.

If the Clipboard contains entities copied with the CopyClip command, then you are prompted in the command bar:
Select insertion point: *(Pick a point, or enter x,y coordinates.)*

Command Options

Option	Description
Select insertion point	Specifies the insertion point for the entities pasted from the Clipboard. Pick a point, or enter x,y coordinates.

Procedures

To paste a selection set as a block
Paste a selection as a block in the target drawing
Move a selection as a block

Related Commands

CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

PasteClip

Pastes entities from the Clipboard into the drawing (short for "paste clipboard").

Accessing the Command

command bar: *pasteclip*
shortcut: *Ctrl+V*
menu bar: *Edit | Paste*
toolbar: *Standard |*

: ***pasteclip***

If the Clipboard contains:

- BricsCAD entities (including BricsCAD text), prompts you in the command bar:
  Select insertion point: (Pick a point in the drawing, or specify x,y coordinates.)
- Non-BricsCAD text, it is immediately pasted in the command bar, and BricsCAD attempts to execute the text as commands.
- Non-BricsCAD entities (such as raster images), they are pasted as OLE entities in the upper-left corner of the viewport.

Command Options

Option	Description
Select insertion point	Specifies the insertion point for the entity.

Procedures

Using Copy and Paste to copy entities between drawings
Using Copybase and Paste to copy entities between drawings
Using Cut and Paste to move entities between drawings

Related Commands

ChSpace - moves entities from paper space to model space and vice versa.
CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteBlock - pastes entities from the Clipboard as a block at an insertion point.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

PasteOrig

Pastes entities from the Clipboard into another drawing, using the coordinates of the source drawing.

Accessing the Command

command bar: *pasteorig*
alias: none
menu bar: *Edit | Paste to original coordinates*

: ***pasteorig***
The content of the Clipboard is pasted into the target drawing.

Procedures

Using Copy and Pasteorig to copy entities between drawings

Related Commands

CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteBlock - pasted entities from the Clipboard as a block at an insertion point.
PasteSpec - displays the Paste Special dialog box for controlling the format of pasted entities.

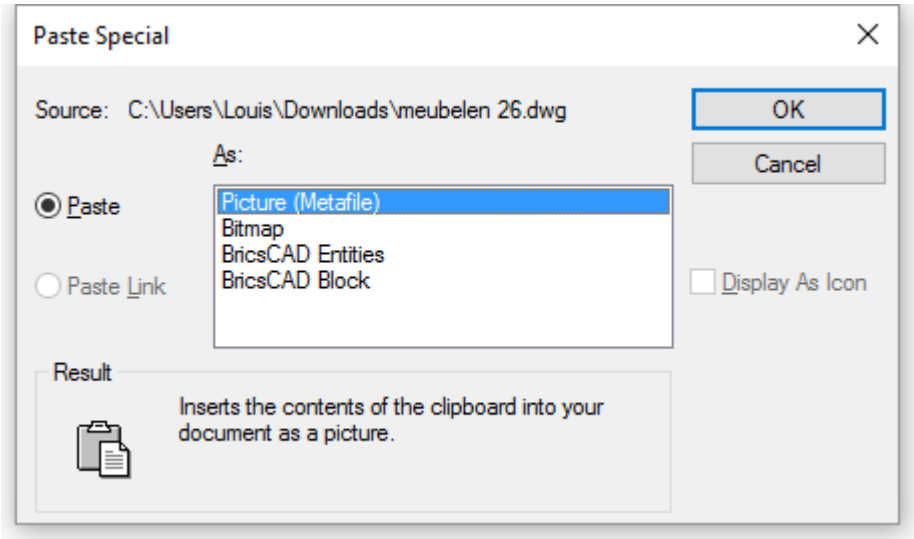
PasteSpec

Pastes entities from the Clipboard through a dialog box (short for "paste special").

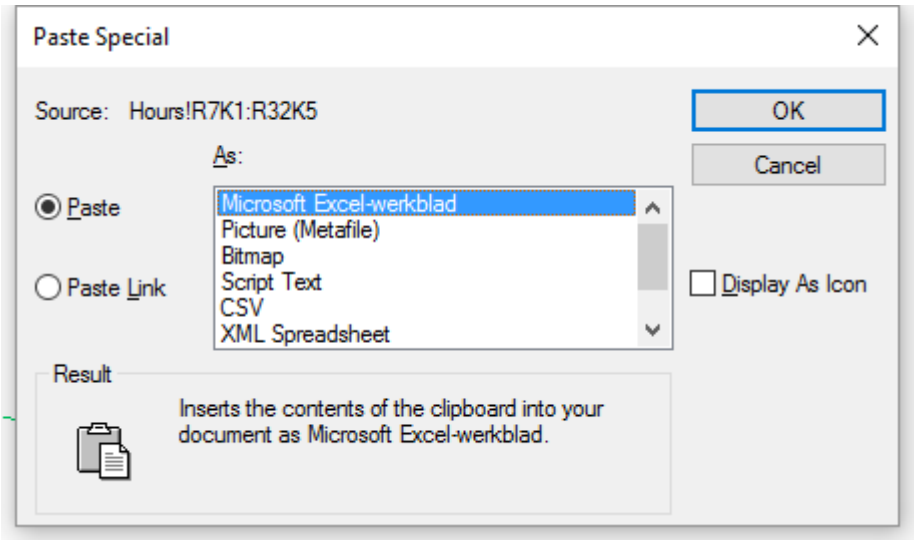
Accessing the Command

command bar: *pastespec*
alias: *pa*
menu bar: *Edit | Paste Special*

: ***pastespec***
Displays a dialog box:



Paste Special dialog when pasting drawing entities



Paste Special dialog when pasting data from an Excel spreadsheet

*Choose a format, and then click **OK**.*

Command Options

Option	Description
Paste	Pastes the entities in the drawing. If the entities are in BricsCAD format, you are prompted: **Select insertion point** - specify the insertion point of the entities. If the entities are in other formats, the entities are pasted in the upper left corner of the viewport.
Paste Link	Pastes the entities in the drawing, and creates a link to the originating application.
As	Chooses the format in which to paste the entities. This list varies, depending on the content of the Clipboard.
Display as icon	Displays the pasted entities by their source icon.

Related Commands

CopyBase- copies selected entities to the Clipboard with a base point.
CopyClip - copies selected entities to the Clipboard.
CopyHist - copies all text from the Text window to the Clipboard.
CutClip - cuts (moves) selected entities to the Clipboard.
PasteBlock - pasted entities from the Clipboard as a block at an insertion point.
PasteClip - pastes entities from the Clipboard into the current drawing.
PasteOrig - pastes entities from the Clipboard at the original coordinates.
Table - creates a table entity in the drawing through a dialog box.

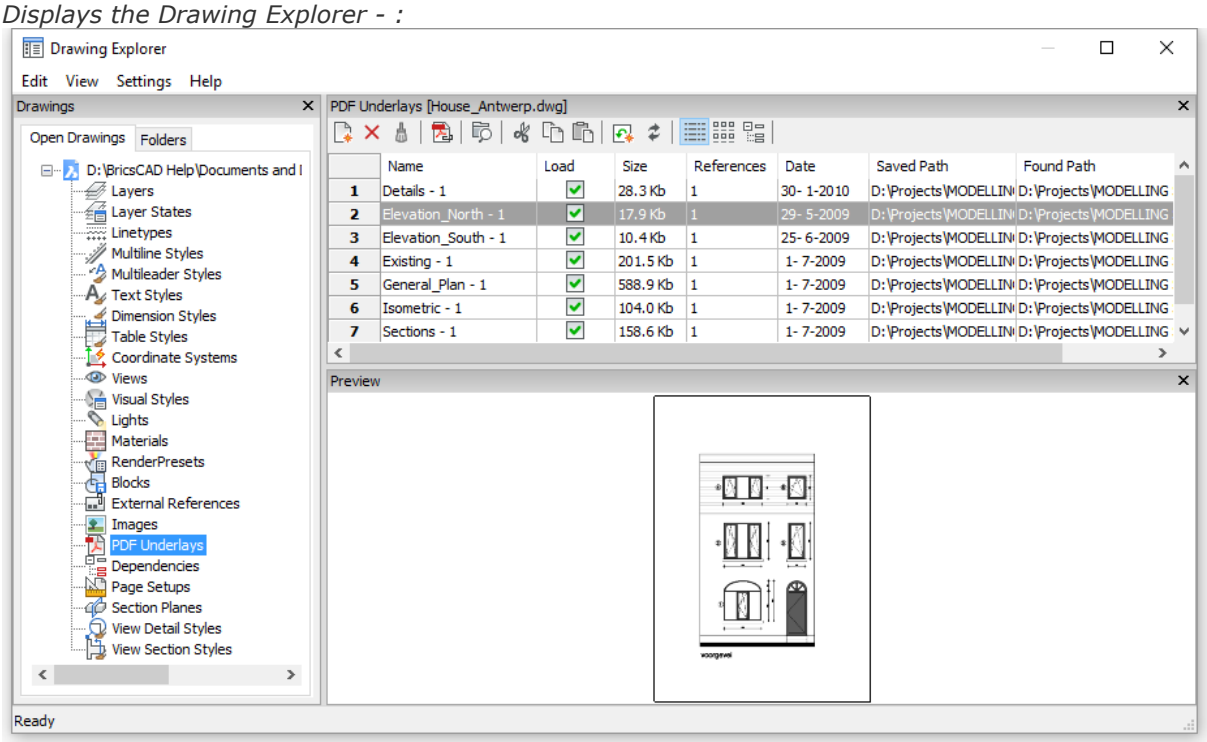
Pdf

Attaches PDF files to the current drawing through the Drawing Explorer.

Accessing the Command

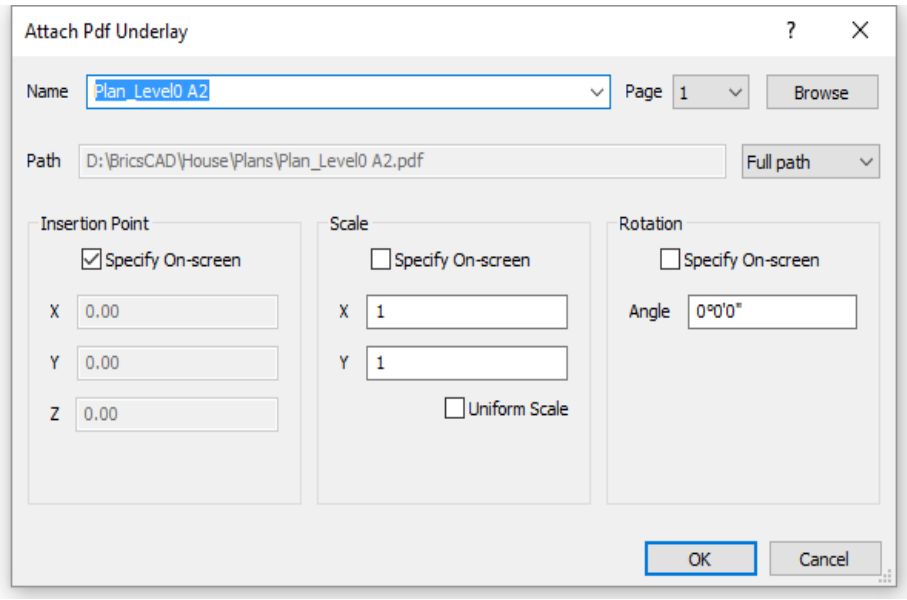
command bar: *pdf*
menu bar: *Insert | Attach PDF...*
toolbar: *Insert |*

: ***pdf***
Displays the Drawing Explorer - :



Command Options

Option	Description
Name	Name of the Pdf insert. This name is assigned by BricsCAD, but cannot be changed by the user.
Loaded	Toggles the loading of the Pdf: • **On** - loads the Pdf, and makes it visible. • **Off** - unloads the Pdf, making it invisible. Click to change the load status.
Size	Reports the size of the PDF files.
References	Reports the number of times the PDF is inserted in the drawing.
Date	Reports the date of the file. This can be useful in determining if you ware working with the most recent revision.

Saved Path	Reports the original path to the PDF file when first loaded. If the path cannot be found, click this field, and then click the button. BricsCAD displays the Choose a File dialog box" 1. Locate the missing DWG file. 2. Click **Open**.
Found Path	Reports the current path to the PDF file.
New	Launches the PDFATTACH command.
Delete	Detaches the currently selected Pdfs
Insert	Inserts another copy of the Pdf. Displays the Attach Pdf Underlay dialog box. 
Detail View	Displays the details of Pdfs.
Icon View	Displays an icon for each Pdf.
Tree View	Displays the relationship between the current drawing and the Pdfs in a tree-like view:

Related Commands

PdfAdjust - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.
PdfAttach - inserts a PDF file as an underlay into the current drawing.
Pdfclip - allows clipping of PDF underlays.
Pdflayers - allows to control the display of layers in a PDF underlay.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
-XRef - attaches and controls xrefs through the command bar.
XClip - clips externally-referenced drawings.
Insert - inserts drawings as blocks.

Eplorer - opens the Drawing Explorer dialog box.

Pdfattach

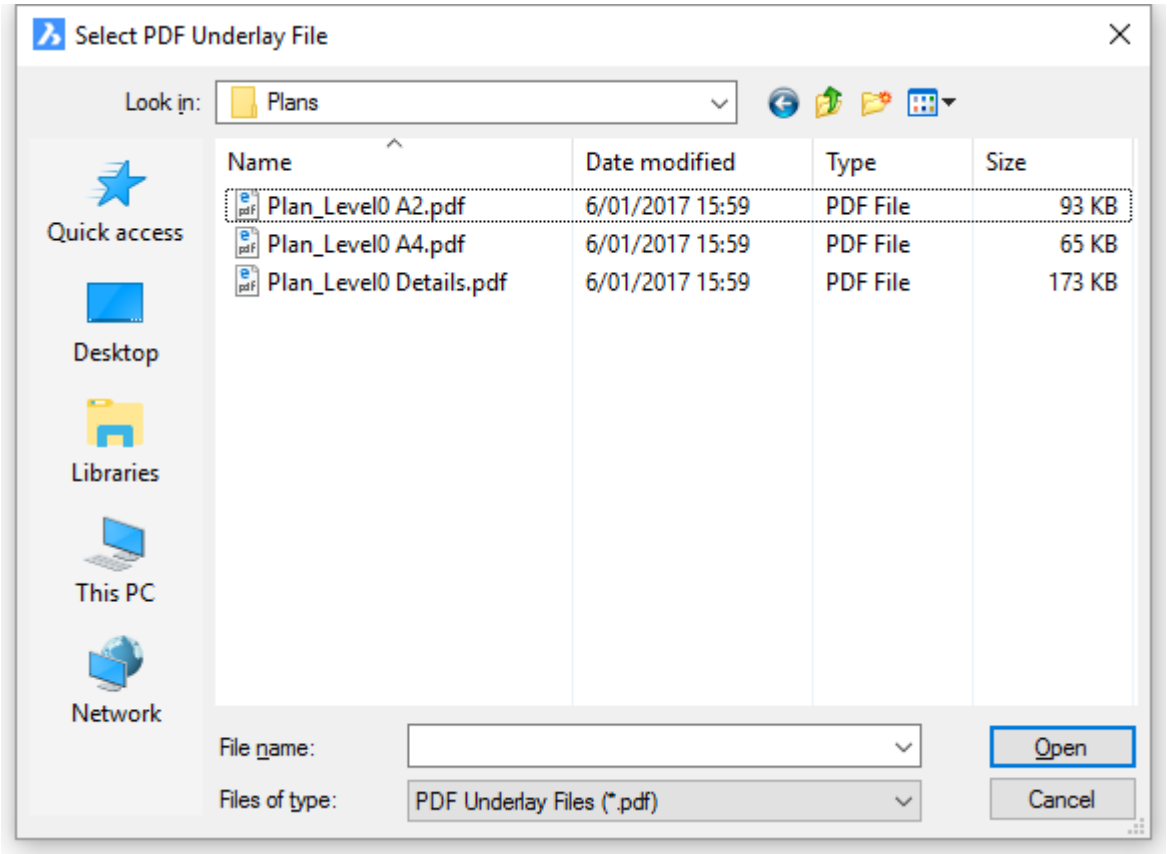
Inserts a PDF file as an underlay into the current drawing; displays dialog boxes.

Accessing the Command

command bar: *pdfattach*
menu bar: *Insert | Attach Pdf...*
toolbar: *Insert |*

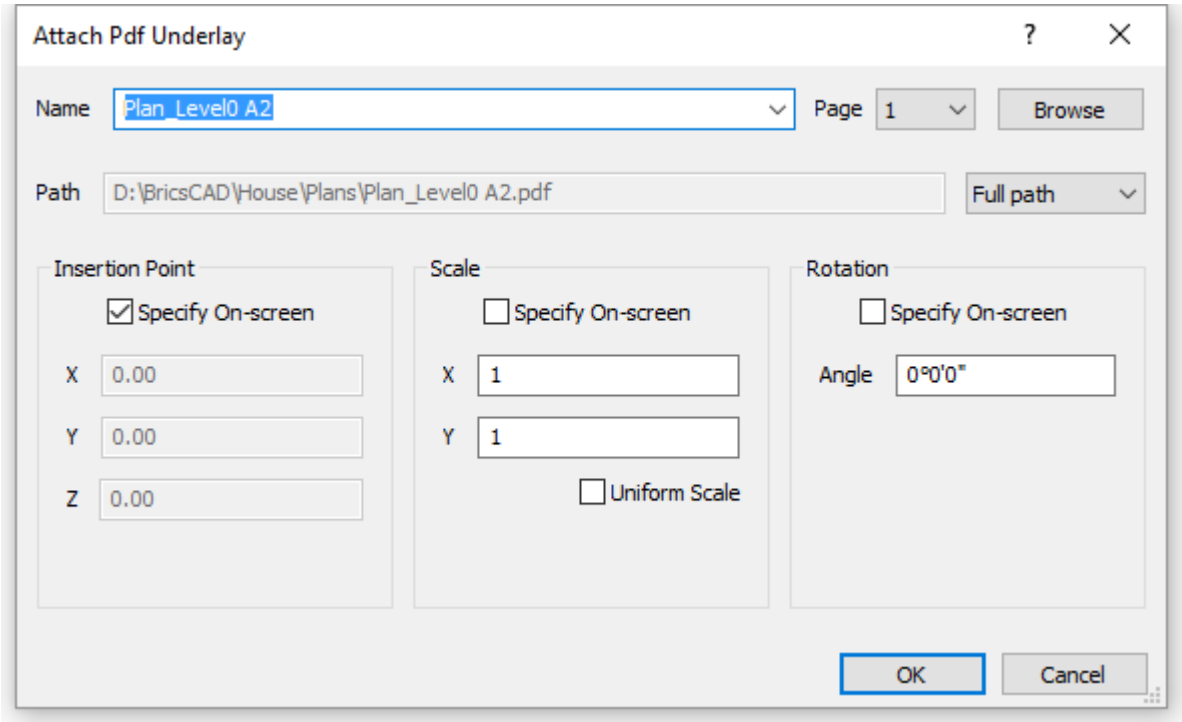
: pdfattach

Displays the *Select PDF Underlay* dialog window.

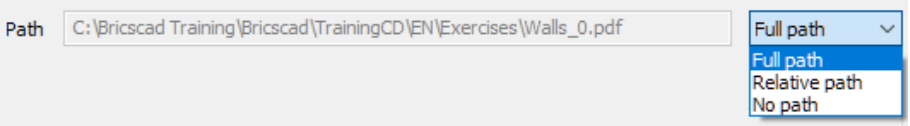


Select a PDF file, then click Open.

BricsCAD displays a dialog box:



Command Options

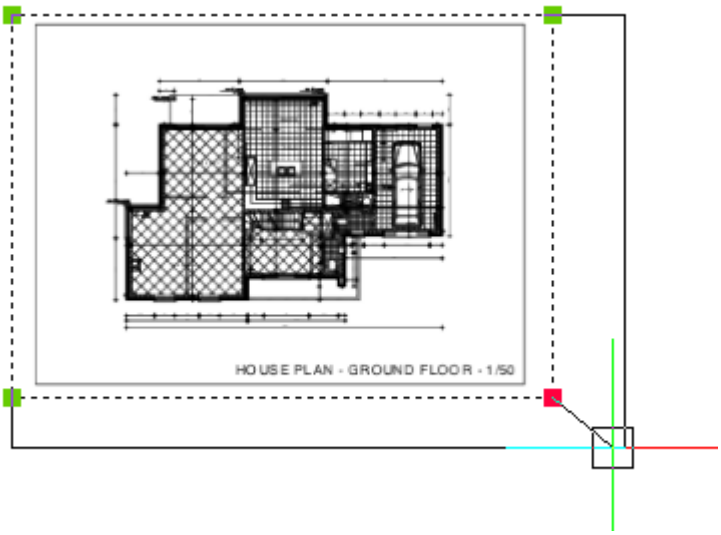
Option	Description
Name	Specifies the name of the attached PDF file. If more than one PDF file has been attached to the drawing, their names are listed by the droplist.
Browse	Displays the *Select PDF Underlay* dialog window. Choose a PDF file, and then click Open.
Path	Click the droplist to choose a different path type:  • Full path - stores the drive and folder(s) names to the PDFfile (absolute reference), such as: c:\cad\PDF\filename.pdf • Relative path - stores the route to the referenced 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(s) names, leaving just the PDF file name, such as: filename.pdf.
Insertion Point	Specifies the location of the underlay's lower-left corner.

Specify On-screen	Determines when the insertion point is specified: • On - specify the insertion point in the drawing after the dialog box is closed; you will be prompted: Insertion point for block - pick a point or enter x,y,z coordinates. • Off - specify the insertion point in the dialog box using the X, Y, and Z fields.
X Y Z	Specifies the x,y,z coordinates for the underlay's insertion point. Use 0,0,0 to insert the underlay at the drawing's origin.
Scale	Specifies the size of the PDF underlay.
Specify On-screen	Determines when the scale factors are specified: • On - specify the scale factor in the drawing after the dialog box is closed; you will be prompted: Scale factor <1.00>: - type a value or specify a point. • Off - specify the scale factor in the dialog box using the X, Y, and Z fields.
X Y Z	Specifies the scale factors of the underlay in the x, y, and z directions. Use 1,1,1 to keep the underlay at its original size.
Rotation	Specifies the rotation angle of the PDF underlay.
Specify On-screen	Determines when the rotation angle is specified: • On - specify the angle in the drawing after the dialog box is closed; you will be prompted: Rotation angle <0.00>: - enter a number, pick two points, or press Enter for the default value. • Off - specify the angle in the dialog box using the Angle field.
Angle	Specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Use 0 to keep the underlay at it original orientation.

Grips Editing

The scaling of a PDF underlay can be edited directly through grips.

1. Select the PDF underlay.
   It has four grips.



2. Drag a grip to adjust the scaling of the PDF underlay.

Procedures

To control the PDF underlay caching
To attach a PDF underlay
To insert a PDF underlay
To control the display of a PDF underlay
To set the layer display in a PDF underlay
To clip a PDF underlay

Related Commands

PdfAdjust - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Pdfclip - allows clipping of PDF underlays.
Pdflayers - allows to control the display of layers in a PDF underlay.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
-XRef - attaches and controls xrefs through the command bar.
XClip - clips externally-referenced drawings.
Insert - inserts drawings as blocks.
-Insert - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.

-PdfAttach

Inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.

Accessing the Command

command bar: *-pdfattach*

: ***-pdfattach***
PDF Underlay file: (Type the path and name of the pdf document in the command bar.)

PDF Underlay page number <1>: (Type the page number or press Enter to accept the current page number.)

Insertion point: (Specify the insertion point in the drawing.)

Scale [Size/XY scales] <1.000000>: (To set a uniform scale factor: either type a value in the command bar or specify a point in the drawing.)

Rotation angle <0.00>: (To set the rotation angle: either type a value in the command bar or specify a point in the drawing.)

Command Options

Option	Description
Size	Sets the size of the insert dynamically; prompts you: Specify opposite corner - Pick a point.
XY Scale factors	Allows to set the scaling in X- and Y direction differently; prompts you: Specify X scale factor <1.000000> - type a value or pick a point. Specify Y scale factor < Equal to X scale factor <1.000000> - so one of the following: • Press enter to set the Y scale factor equal to the X scale factor. • Specify a point in the drawing. • Type a value.

Related Commands

PdfAdjust - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.
Pdfclip - allows clipping of PDF underlays.
Pdflayers - allows to control the display of layers in a PDF underlay.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
-XRef - attaches and controls xrefs through the command bar.
XClip - clips externally-referenced drawings.
Insert - inserts drawings as blocks.
-Insert - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.

PdfAdjust

Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.

Accessing the Command

command bar: *pdfadjust*
menu: *Modify | PDF Underlay | Adjust PDF*

: ***pdfadjust***
Prompts you in the command bar:

Select PDF underlay: (Select a pdf underlay in the drawing.)
Entities in set: 1

Select PDF underlay: (Right click or select another pdf underlay in the drawing.)

Enter PDF underlay option [Fade/Contrast/Monochrome] <Fade>: (Enter an option.)

Command Options

Option	Description
Fade	Sets the *Fade* setting of a PDF underlay. Prompts you: **Enter fade value [0-100] <0>**: Type a value.
Contrast	Sets the *Contrast* setting of a PDF underlay. Prompts you: Enter contrast value [0-100] <100>: Type a value.
Monochrome	Sets the *Monochrome* setting of a PDf underlay. Prompts you: Monochrome? <Yes>/No: Type *Y* or *N*.

Procedures

To adjust the display settings of a PDF underlay

Related Commands

Pdf - attaches PDF files to the current drawing through the Drawing Explorer.
Pdfattach - inserts a PDF file as an underlay into the current drawing; displays dialog boxes.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Pdfclip - clips a PDF underlay.
Pdflayers - allows to control the display of layers in a PDF underlay.

Pdfclip

Clips a PDF underlay.

Accessing the Command

command bar: *pdfclip*

: pdfclip

Select PDF underlay: (Choose a pdf underlay.)

Enter PDF clipping option [ON/OFF/Delete/New] <New>: (Enter an option, or press Enter for the default.)

Command Options

Option	Description
Select PDF underlay	Selects the PDF underlay to clip. Click the frame of the PDF underlay to select.
Delete	Deletes the existing clipping boundary.
New	Depending on whether or not a boundary already exits the command will produce following prompts: If no boundary exists you are prompted: Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>: - type P or R. If a boundary already exists you are prompted: Delete old boundary? [Yes/No] - type Y or N. This option replaces the existing clipping boundary with a new one. Please note that the command will only continue (by prompting you to select a new boundary) after the previous boundary is deleted. So typing No at previous prompt will cancel the Pdfclip command.
Polygonal	Creates multi-sided clipping boundaries; prompts you: Start of polyline - pick a point in the drawing, or specify x,y coordinates. Next point - pick another point in the drawing, or specify x,y coordinates. Undo/next point - pick another point in the drawing, or specify x,y coordinates. Undo/next point - pick more points, or press Enter to complete the polygon. Type U to undo the previous point.
Rectangular	Creates a rectangular clipping boundary; prompts you: **Specify first corner** - pick a point in the drawing, or specify x,y coordinates. **Specify opposite corner** - pick another point in the drawing, or specify x,y coordinates.
ON	Turns the clipping boundary on.
OFF	Turns the clipping boundary off; all of the PDF underlay is again visible. The clipping boundary is saved though.

Procedures

To clip a PDF underlay
To toggle the display of the clipped part of a PDF underlay
To delete a clipping boundary

Related Commands

ImageClip - clips images.
PdfAdjust - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.
pdfattach - inserts a PDF file as an underlay into the current drawing.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Pdflayers - allows to control the display of layers in a PDF underlay.
VpClip - clips viewports.
XClip - clips externally-referenced drawings.

Pdflayers

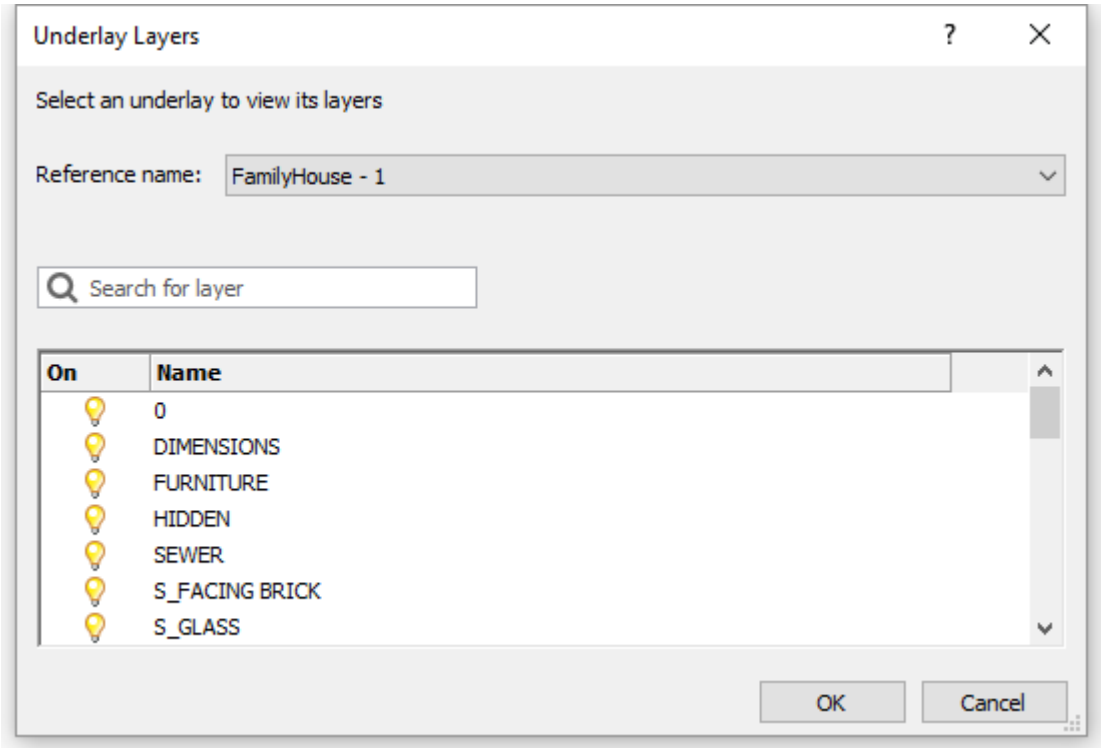
Allows to control the display of layers in a PDF underlay.

Accessing the Command

command bar: *pdflayers*

: ***pdflayers***
Prompts you in the command bar:
Select PDF underlay: *(Click the frame of a pdf underlay.)*

BricsCAD displays a dialog box:



Command Options

Option	Description
Reference name	Select a different PDF underlay.
Search for layer	Type a search string in the search field. Only layers of which part of the layer name matches the search string are listed. You can use wildcards (? or *) in the search field.
On	Click the column title to sort the layer list by display.
or	Click the icon to turn the display of a layer on/off.
Name	Click the column title to sort the layer list by name.

Procedures

To set the layer display in a PDF underlay

Related Commands

PdfAdjust - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar.
pdfattach - inserts a PDF file as an underlay into the current drawing.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
pdfclip - clips a PDF underlay.

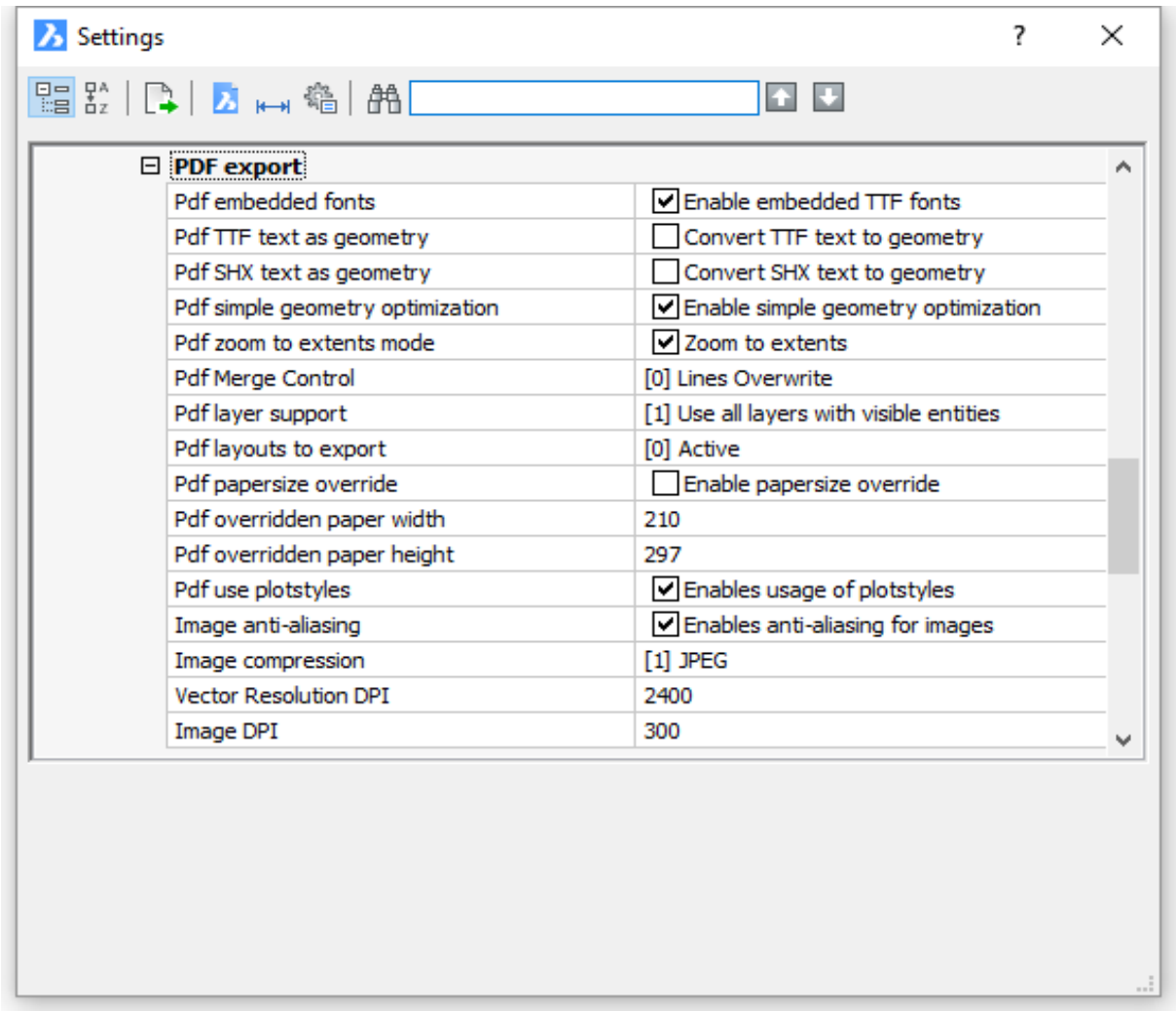
PdfOptions

Controls the exporting of drawing in PDF format through the Settings dialog box.

Accessing the Command

command bar: *pdfoptions*
menu bar: *File | PDF Export Options*

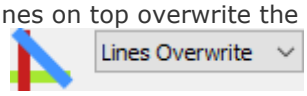
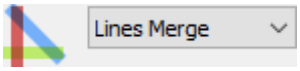
: ***pdfoptions***
Displays the PDF Export section of the Settings dialog box:



Choose options, and then click the X to exit the dialog box.

To export drawings in PDF format, use the Export command.

Command Options

Option	Description
Pdf embedded fonts	Determines whether TTF font files are embedded in the PDF file: • Yes - includes fonts, which may violate the copyright on some fonts. • No - does not include fonts, which may result in missing fonts if the receiving computer does not have the required SHX or TTF files.
Pdf TTF text as geometry	Determines whether TTF (true type fonts) are exported as geometry: • Yes - explodes fonts into lines and arcs. • No - does not explode fonts.
Pdf SHX text as geometry	Determines whether SHX (shapes compiled) are exported as geometry: • Yes - explodes font into lines and arcs. • No - does not explode fonts.
Pdf simple geometry optimization	Determines if the geometry is simplified to reduce file size: • Yes - optimizes geometry, such as turning multiple segments into a single polyline. • No - does not optimize geometry.
Pdf zoom to extents mode	Determines if the drawing is zoomed to extents: • Yes - zooms drawing to extents, which ensures the drawing matches the paper size. • No - does not zoom the drawing, which ensures the drawing is exported as-is.
Pdf Merge Control	Controls the appearance of crossing lines. • Lines Overwrite - the lines on top overwrite the lines below. See the Publish dialog:  • Lines Merge - the colors of crossing lines are blended into a new color. See the Publish dialog: 
Pdf layer support	Determines how layers are exported: • Don't use layers - removes all layer information. • Use layers/Don't include OFF-layers - excludes layers that are turned off. • Use layers/Include OFF-layers - exports all layers. See **Layer** command.
Pdf layouts to export	Determines which layouts to export: • Active - exports the current layout. • All - exports all layouts. Each layout is plotted to a separate page in the PDF file.
Pdf papersize override	Determines whether page size is specified: • Yes - uses the custom page size specified below. • No - does not use specified page size.

Pdf overridden paper width	Specifies the width of the page, in mm or inches.
Pdf overridden paper height	Specifies the height of the page, in mm or inches.
Pdf use plotstyles	Use plotstyles when exporting to pdf.
Image Antialiasing	Enables anti-aliasing for images that require upscaling when exporting.
Image Compression	Sets the compressing for exported images.
Vector Resolution DPI	Sets the resolution for vector graphics when exporting. Values between 72 and 40000 are accepted.
Image DPI	Controls the minimum resolution for exported raster images. The value cannot exceed the value set for Vector Resolution DPI.

Related Commands

Export - exports drawings in PDF format.
Plot and Print - export drawings to PDF files if the *Print As PDF.pc3* printer configuration is selected.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

PEdit

Edits polylines, 3D polylines, and 3D meshes (short for "polyline edit").

Accessing the Command

command bar: *pedit*
aliases: *editpline, pe*
menu: *Modify | Edit Polyline*
toolbar: *Modify |*
mouse: (double click a polyline)

: ***pedit***
Prompts you in the command bar:

Select polyline to edit [Multiple]: (Choose one polyline or polyface mesh or type M to select multiple polylines.)

When a 2D polyline is selected, you are prompted:

Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: (Enter an option, or press Enter to exit the command.)

When a 3D polyline is selected, you are prompted:

Edit polyline [Edit vertices/Close/Decurve/Join/Reverse/Spline/Undo]<eXit>: (Enter an option, or press Enter to exit the command.)

When a 3D mesh is selected, you are prompted:

Edit polyline: [Edit vertices/Desmooth/Mopen/Nclose/Undo]<eXit>: (Enter an option, or press Enter to exit the command.)

When a line, arc, circle or spline is selected, you are prompted:

The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-Turn into polyline>: (Press Enter to turn the selected entity into a 2D polyline.)

Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: (Enter an option, or press Enter to exit the command.)

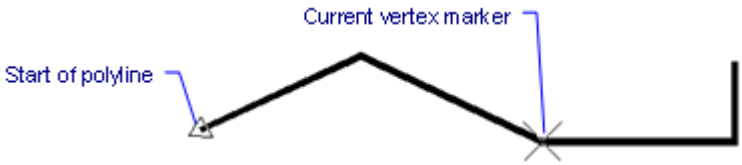
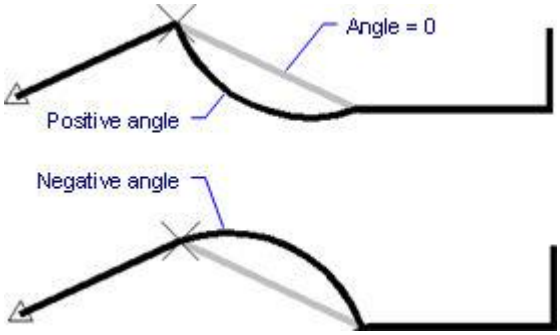
When the Multiple option is chosen, you are prompted:
Select entities: (Select lines, polylines, arcs and/or splines)

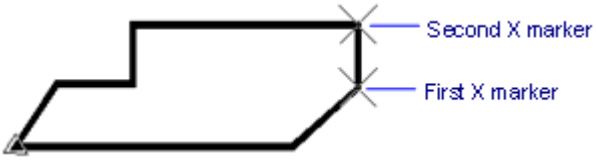
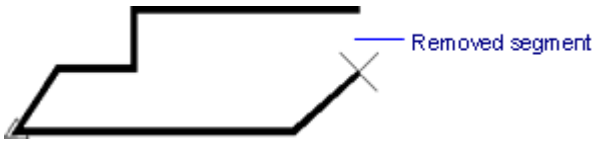
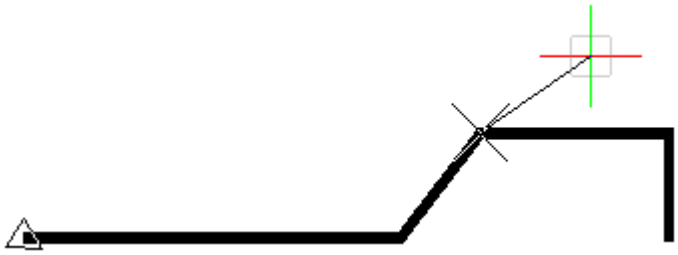
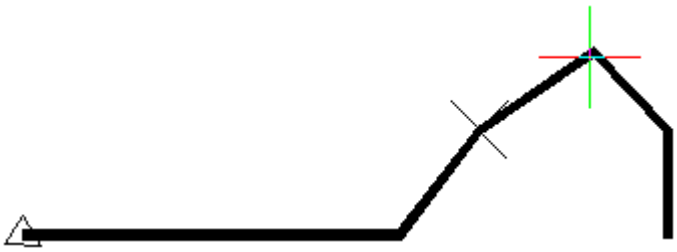
When one or more lines, arcs, circles or spline are selected, you are prompted:

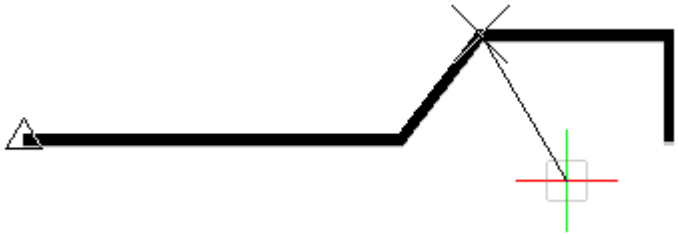
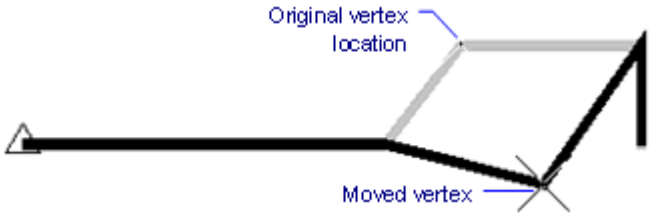
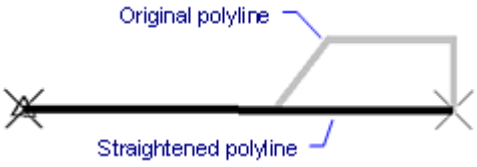
The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-Turn into polyline>: (Press Enter to turn the selected entity into a 2D polyline.)

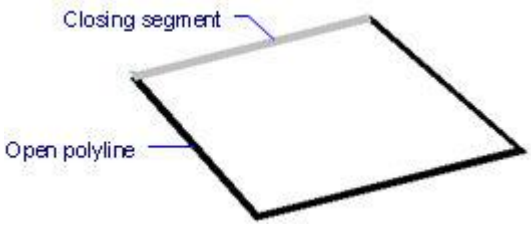
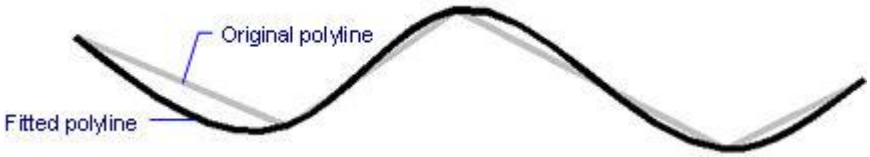
Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: (Enter an option, or press Enter to exit the command.)

Command Options

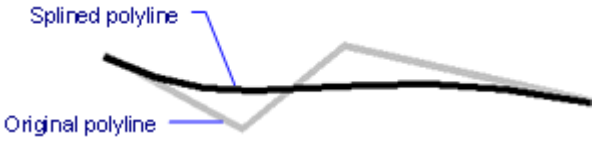
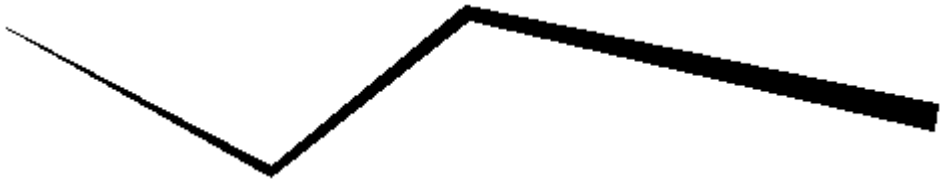
2D Polyline Option	Description
Edit vertices	Edit vertices; prompts you: **Edit vertices:** **Next/Previous/Angle/Break/Insert/Move/Regen/SElect/Straighten/Width/eXit/<Next>** - enter an option. BricsCAD displays a triangle marker at the start of the polyline, and an X marker at the current vertex.  Enter **Next** or **Previous** to move the marker from vertex to vertex. Alternatively, use the SElect option to select the vertex directly.
Next	Moves the X marker to the next vertex. When the marker reaches the last vertex of a closed polyline, it does not move to the "next' (first) vertex.
Previous	Moves the X marker to the previous vertex.
Angle	Changes the angle of the current segment: • If an arc segment, changes the angle (curvature) to the arc. • If a line segment, turns it into an arc. • Enter 0 to convert an arc segment into a line segment.  Prompts you: **Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <0>** - enter an angle.
Break	Removes segments from the polyline; prompts you: **Next/Previous/Select/Go/eXit/<Next>** - enter an option. BricsCAD marks the two vertices that define the segments to be removed with two X markers.

	
Next	Moves the second X marker to the next vertex.
Previous	Moves the second X marker to the previous vertex.
Select	Moves second X marker directly to another vertex; prompts you: **Select point along segment move to** - pick a point on the polyline. BricsCAD places the second X marker on the vertex nearest to the pick point. This option is faster than using Next and Previous.
Go	Erases the segments between two marked vertices.  This action turns: • Closed polylines into open polylines. • Open polylines into two polylines, with a gap between them. This action cannot be undone.
eXit	Exits this sub-sub-option; returns to the previous prompt line.
Insert	Inserts a vertex; prompts you: **Location of vertex** - pick a point for the new vertex. The drag line previews the location of the new vertex.  BricsCAD adds the vertex and segment. 
Move	Moves the current vertex; prompts you: **New location for vertex** - pick point to move the vertex. The drag line previews the new location of the vertex.

	 BricsCAD moves the vertex and associated segments.  It is easier to move vertices using grips.
Regen	Regenerates the polyline to show changes made by the editing actions of this command.
SElect	Moves X marker directly to another vertex; prompts you: **Select point along segment to edit** - pick a point on the polyline. BricsCAD places the X marker on the vertex nearest to the pick point. This option is faster than using the Next and Previous options.
Straighten	Removes all segments between two vertices; prompts you: **Straighten: Next/Previous/Select/Go/eXit** - enter an option When you select the start and end points of an multi-segment polyline, the entire polyline is straightened, becoming a single segment; this also applies to closed polylines.
Next	Moves the second X marker to the next vertex.
Previous	Moves the second X marker to the previous vertex.
Select	Moves second X marker directly to another vertex; prompts you: **Select point along segment move to** - pick a point on the polyline. BricsCAD places the second X marker on the vertex nearest to the pick point. This option is faster than using the Next and Previous options.
Go	Replaces the segments between the two marked vertices with a single segment.  This action cannot be undone.
eXit	Exits this sub-sub-option; returns to the previous prompt line.
Width	Changes the width of the current segment; prompts you: **Enter new starting width** - specifies the width at the starting end of the polyline. **Enter new ending width** - specifies the width at the polyline's end. The "current" segment lays between the X-marked vertex and the next vertex.

eXit	Exits this sub-option; returns to the previous prompt line.
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 the Fit and Spline options.
Fit	Fits a curve to the polyline.  To unfit the polyline, use the Decurve option.
Join	When editing a single polyline: Adds lines, arcs, and polylines to the selected polyline; prompts you: **Select arc or line for joining:** This option works only with open polylines and with cojoined lines, polylines, and arcs; it does not work with traces. Cojoined means that the endpoints of entities meet, such as when drawn with ENDpoint object snap. **The number of already selected entities displays in the command bar:** Entities in set: 3 **Press Enter or right click to create the polyline:** 3 segments added to polyline The joined entities take on the source polyline's properties, such as color, width, and layer. When editing multiple polylines: ; prompts you: **Enter fuzz distance or [Jointype]:** Do one of the following: • Press Enter to create one or more polylines with cojoined lines, polylines, and arcs in the selection set. • Enter a so called 'fuzz distance' then press Enter to create one or more polylines with cojoined lines, polylines, and arcs in the selection set. If endpoints exist within the fuzz distance lines, polylines and arcs are extend or trimmed. • Type **J**, then press Enter; prompts you: **Specify type of joining: [Extend/Add/Both]:** Select a jointype:

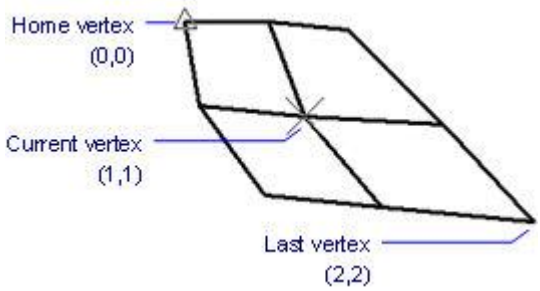
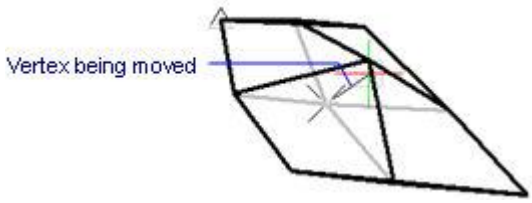
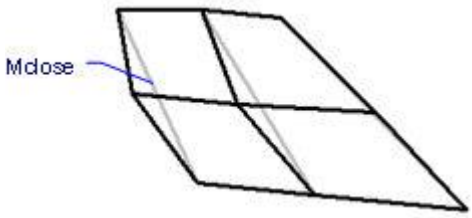
	• **Extend**: Extends or trims the segments to their nearest endpoints that lie within the fuzz distance. • **Add**: Adds linear segments between their nearest endpoints that lie within the fuzz distance.. • **Both**: If possible joins the selected polylines by extending or trimming, otherwise adds linear segments. BricsCAD reports the number of polylines created in the command bar: **#segments added to # polylines**

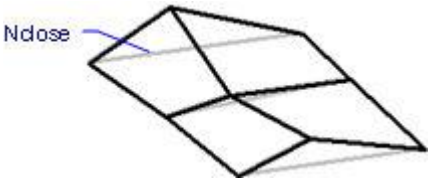
Linetype-mode	Determines how linetype appear on polylines; prompts you: **Linetype continuous along polyline: ON/OFF <OFF>:** • **ON** - linetypes start and stop at the polyline's start and endpoints. • **OFF** - linetypes start and stop at each vertex. (This option is related to the PLineGen system variable.)
Reverse	Reverses the direction of the polyline: the starting point is now its end point, and vice versa. The polyline does not change its look with this operation. This option affects operations that depend on the direction of a polyline, such as vertex editing. The polyline does not change its look with this operation, except that the triangle marker moves to the other end of open polylines.
Spline	Converts the polyline into a spline.  All width information is lost; use the Width option to reapply width. The spline is a Bezier-spline whose smoothing is defined by the SplineType system variable.
Taper	Tapers the width of the entire polyline from one end to the other; prompts you: **Enter new starting polyline width** - specify the width at the starting end of the polyline. **Enter new ending polyline width** - specify the width at the polyline's end. 
Width	Changes the width of all segments; prompts you: **Enter new width for all segments** - specify a width. This option overrides tapered widths.
Undo	Undoes the last action.
eXit	Exits the command.

3D Polyline Option	Description

Edit vertices	Edit vertices; prompts you: **Edit vertices: Next/Previous/Break/Insert/Move/Regen/SElect/Straighten/eXit/<Next >** - enter an option. See polyline options, above. (The Angle and Width options are missing, because 3D polylines cannot contain arcs and cannot have width.)
Close	Closes open 3D polylines by adding a segment between the start and endpoints. When the 3D polyline is closed, this options reads Open.
Open	Opens closed 3D polylines by removing the last segment drawn to close the polyline. When the 3D polyline is open, this options reads Close.
Decurve	Reverses the effects the Fit and Spline options.
Join	Adds lines, arcs, and polylines to the selected polyline; prompts you: **Select entities** - choose lines and arcs. This option works only with open polylines and with cojoined lines, polylines, and arcs; it does not work with traces. Cojoined means that the endpoints of entities meet, such as when drawn with ENDpoint object snap. The joined entities take on the source polyline's properties, such as color, width, and layer.
Reverse	Reverses the direction of the 3D polyline: the starting point is now its end point, and vice versa. The 3D polyline does not change its look with this operation. This option affects operations that depend on the direction of a polyline, such as vertex editing. The polyline does not change its look with this operation, except that the triangle marker moves to the other end of open polylines.
Undo	Undoes the last action.
Exit	Exits the command.

3D Polyface Mesh Option	**Description**
Edit vertices	Moves vertices; prompts you: **Edit vertex (0,0): Select/Next/Previous/Left/RIght/Up/Down/Move/REgen/eXit** - enter an option. BricsCAD displays a triangle marker on the home vertex (0,0) and an X marker on the current vertex.

	 It is easier to move vertices using grips.
Select	Moves X marker directly to another vertex; prompts you: **Select point along segment to edit** - pick a point on the mesh. BricsCAD places the X marker on the vertex nearest to the pick point. This option is faster than using the Next and Previous options.
Next	Moves the X marker to the next vertex. When the marker reaches the last vertex, it does not move to the "next' (first) vertex.
Previous	Moves the X marker to the previous vertex.
Left	Moves the X marker to the vertex on the left.
RIght	Moves the X marker to the vertex on the right.
Up	Moves the X marker to the higher-numbered vertex.
Down	Moves the X marker to the lower-numbered vertex.
Move	Moves the current vertex; prompts you: **New location for vertex** - pick a point to move the vertex. The drag line previews the new location of the vertex.  BricsCAD moves the vertex and associated faces.
REgen	Regenerates the polyline to show changes made by the editing actions of this command.
eXit	Exits this sub-option; returns to the previous prompt line.
Desmooth	Removes the Bezier-spline smoothing of 3D meshes.
Mclose	Closes the 3D mesh in the m direction:  When the mesh is closed, this option reads MOpen.

Mopen	Opens the 3D mesh in the m direction. When the mesh is closed, this option reads MClose.
Nclose	Closes the 3D mesh in the n direction:  When the mesh is closed, this option reads NOpen.
Nopen	Open the 3D mesh in the n direction. When the mesh is closed, this option reads NClose.
Undo	Undoes the last action.
eXit	Exits the command.

Procedures

Converting an entity into a polyline
To fit a curve to a polyline
Convert a straight polyline segment into an arc
Break a polyline into two separate polylines
Insert a new vertex in a polyline
Move a vertex in a polyline
Delete vertices in a polyline
Change the width of a polyline segment
To apply a uniform width to a polyline
To taper a polyline uniformly along its length
To join an arc, line, or polyline to an existing polyline
Opening and closing polylines
Setting the Linetype generation mode

Related Commands

3dPoly - draws 3D polylines.
3dFace - draws 3D 4-edged faces with optional invisible edges.
3dMesh - draws 3D surface meshes of size m x n.
ConvertPoly - converts lightweight polylines to classic polylines (2D polylines) and vice versa.
Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
PeditExt - Edits vertices and segments of a polyline.
PFace - draws 3D multi-sided meshes.
PLine - draws polylines.
Splinedit - modifies the properties of a spline or converses a spline to a polyline.

PeditExt

Edits vertices and segments of a polyline.

Accessing the Command

command bar: *peditext*
Quad cursor menu: *2D Editing* | | | | | | |

: ***peditext***
Prompts you in the command bar:

Select option [Add vertex/add vertex at End/adjust Bulge/Remove segment/Stretch segment/stretch Vertex]: (Select an option)

Command Options

Option	Description - command line	Quad Icon	Description - Quad cursor menu
Add vertex	Adds a vertex to the selected segment. Prompts you: Select the polyline segment to add a vertex to - select a polyline segment The new vertex displays dynamically. Select a point - specify the location of the new vertex.		Adds a vertex to the segment under the cursor. The new vertex displays dynamically. Prompts you: Select a point - specify the location of the new vertex.
Add vertex at end	Adds a new vertex at the end of the polyline. Select the polyline near the end you wish to add a vertex - click a polyline near the end point The new vertex displays dynamically. Select a point - specify the location of the new vertex.		Adds a new vertex at the end of the polyline. This option is only available if the cursor is on one of the endpoints of a polyline. The new vertex displays dynamically. Prompts you: Select a point - specify the location of the new vertex.
Adjust bulge	Modifies the bulge factor* of the selected segment. Prompts you: Select the polyline segment to adjust bulge - select a polyline segment The bulge of the selected segment changes dynamically. Select a point - specify a point to set the new bulge.		Modifies the bulge factor* of the segment under the cursor. The bulge of the segment changes dynamically. Prompts you: Select a point - specify a point to set the new bulge.
Remove segment	Deletes the selected segment. Prompts you: Select the polyline segment to remove - select a polyline segment		Deletes the segment under the cursor. The vertex closest to the selection point is removed.

Stretch segment	Moves the selected segment. Prompts you: Select the polyline segment to stretch - select a polyline segment The selected segment moves dynamically; the adjacent segments are stretched. Select a point - specify the location of the segment		Moves the segment under the cursor. The segment moves dynamically; the adjacent segments are stretched. Prompts you: Select a point - specify the location of the segment
Stretch vertex	Moves the selected vertex. Prompts you: Select the polyline vertex to stretch - select a polyline vertex The selected vertex moves dynamically. Select a point - specify the location of the vertex		Moves the vertex under the cursor. The vertex moves dynamically. Prompts you: Select a point - specify the location of the vertex

(*) Bulge factor: defines whether a polyline segment is straight (bulge = 0) or arched (bulge > 0: counter clockwise; bulge < 0: clockwise)

Procedures

Editing a polyline using the Quad

Related Commands

PEdit - edits polylines, 3D polylines, and 3D meshes.

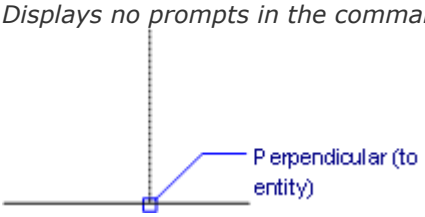
Perpendicular

Toggles perpendicular entity snap mode.

Accessing the Command

command bar: *perpendicular*
transparent: *per*
toolbar: *Entity Snaps* |

: ***perpendicular***
Displays no prompts in the command bar.



When perpendicular entity snap is on, the cursor snaps to points of perpendicularity to entities.

Command Options

Option	Description
Perpendicular	Entering this command toggles Perpendicular entity snap. It snaps to points of perpendicularity. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Quadrant - toggles quadrant snap mode.
Tangent - toggles tangent snap mode.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.

PFace

Draws 3D multi-sided meshes; meant for use by programs (short for "polyface mesh").

Accessing the Command

command bar: *pface*
menu: *Model | Meshes | Polyface Mesh*
toolbar: *Meshes | 3D Faces |*

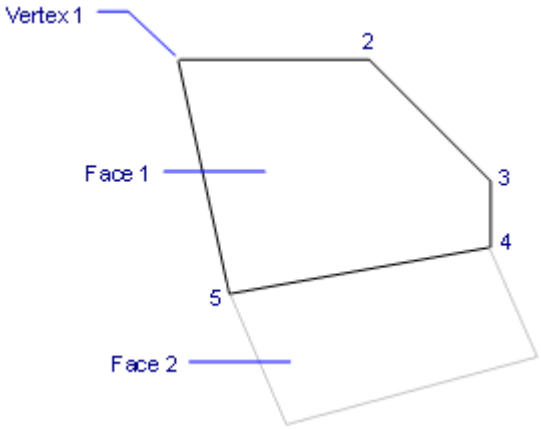
: ***pface***
Prompts you in the command bar:

Position of mesh vertex 1: (Pick a point, or enter x,y,z coordinates.)

Position of mesh vertex 2/<Enter to define faces>: (Pick a point, or enter x,y,z coordinates.)

Position of mesh vertex 3/<Enter to define faces>: (Pick a point, enter x,y,z coordinates, or press Enter to stop drawing polyfaces.)

Face 1, Vertex 1: Color/Layer/<Exit>: (Enter a number, or an option.)



Command Options

Option	Description
Position of mesh vertex 1:	Specifies the first vertex of the face. A face can have up to 256 sides.
Position of mesh vertex n	Specifies the next vertex of the face.
Color	Specifies the color of the edge; see **Color** command.
Layer	Specifies the layer of the edge; see **Layer** command.
Face n, vertex n	Specifies the face and vertex number. Enter a negative number to make the edge invisible.

Related Commands

3dFace - draws 3D 4-edged faces with optional invisible edges.
3dMesh - draws 3D surface meshes of size m x n.
PEdit - edits 3D meshes.

PlaceView

Places a model view selected from a source drawing into a paper space layout of the current drawing.
The command applies to paper space only.

Accessing the Command

Drag a model view from the Content Browser panel onto a paper space layout of the current drawing.
or
Command bar: *placeview*

: placeview
Prompts you in the command bar:
Enter file name: (Type the drawing name, including the full path.)
Enter view name to place or [?]: (Type the name of the model view.)
Enter view origin point: (Specify a point.)
The selected drawing is attached as an external reference and a paper space viewport showing the selected model view is placed in the layout.

Command Options

Option	Description
?	Prints a list of the saved model views in the Prompt History window.

Related Commands

ContentBrowserOpen - opens the Content Browser panel.
XAttach - attaches externally-referenced drawings through a dialog box.
XRef - attaches DWG files to the current drawing through the Drawing Explorer .

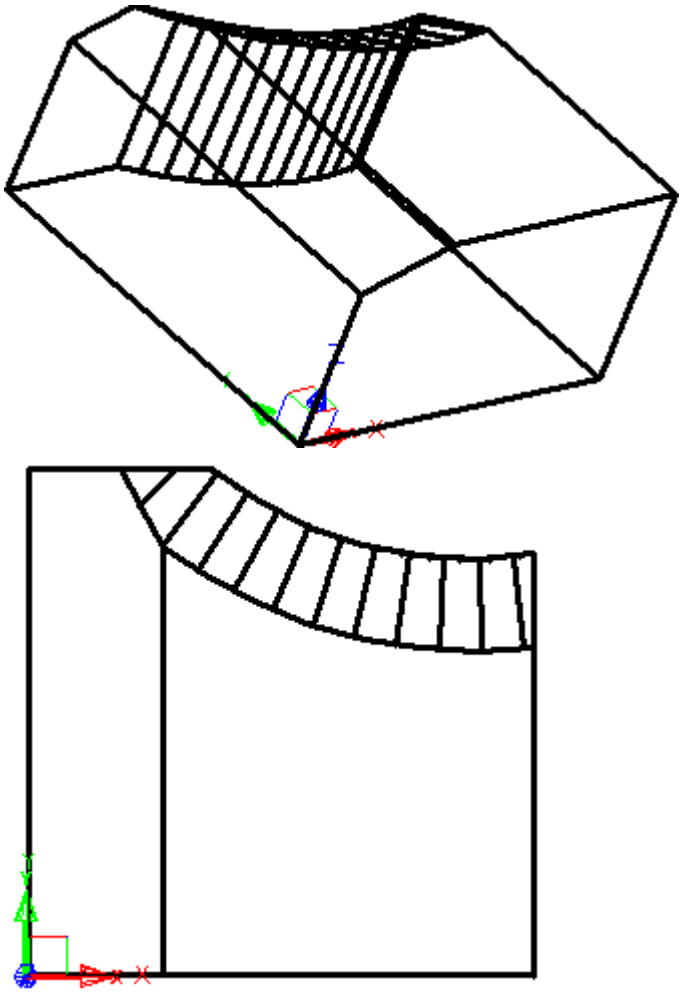
Plan

Displays the plan viewpoint of drawings.

Accessing the Command

command bar: *plan*
menu bar: *View | Plan View*
toolbar: *View |*

: ***plan***
Prompts you in the command bar:
Plan view of [Current/Ucs/World] <Current UCS>: *(Enter an option, or press Enter.)*



Left: 3D model in 3D viewpoint; right: 3D model in plan view.

Command Options

Option	Description
UCS	Displays the plan view of a named UCS; prompts you: **? to list/<Name of UCS>** - specify the name of the UCS.
World	Displays the plan view of the World Coordinate System.
Current	Displays the plan view of the current UCS.

Related Commands

SetUcs - sets the UCS to a specified viewpoint.
UcsIcon - controls the appearance of the UCS icon.
Ucs - creates and modifies user-defined coordinate systems.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
Settings - sets parameters of UCSs.

PLine

Draws polyline lines, arcs, and splines (short for "polyline"). A polyline is a multi-segment entity that can contain lines, arcs, and have width.

Accessing the Command

command bar: *pline*
aliases: *pl, polyline*
menu bar: *Draw | Polyline*
toolbar: *Draw |*

: ***pline***
Prompts you in the command bar:

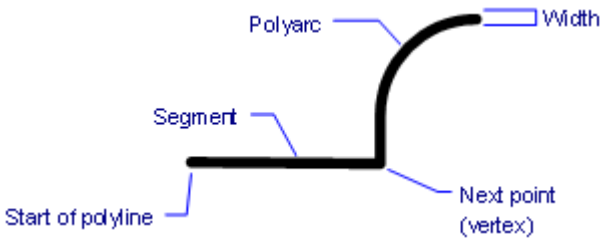
Start of polyline: (Pick a point.)
or

Select start of polyline or [Follow] <Last point>: (Pick a point or enter an option.)

Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: (Pick a point, or enter an option.)

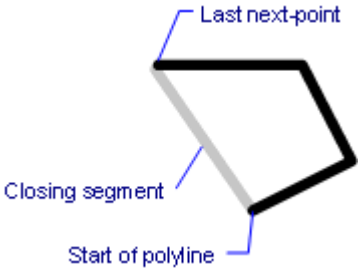
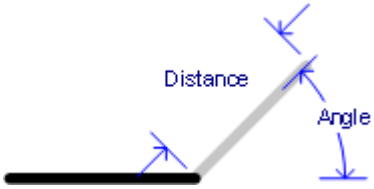
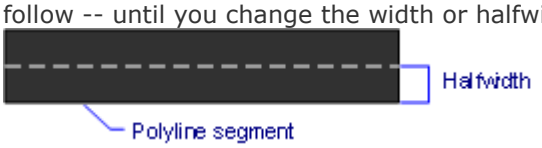
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: (Pick a point, enter an option, or press Enter to exit.)

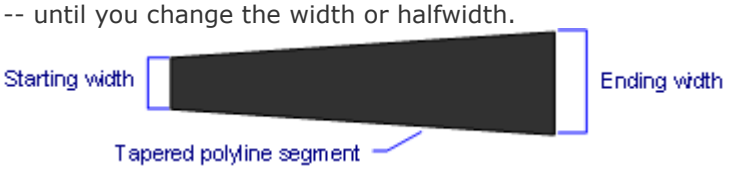
Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]: (Pick a point, enter an option, or press Enter to exit.)



Command Options

Option	Description
Start of polyline	Specifies the start point of the polyline's first segment. Enter x,y,z coordinates, or pick a point in the drawing.
Last point	Starts the polyline at the last point picked in the drawing.
Next point	Specifies the next point (endpoint for the current segment). Each next-point becomes a vertex in the polyline.
Arc	Switches to arc-drawing mode; draws arc segments. Prompts you: Set end of arc or [draw Lines/Angle/CEnter/CLose/Direction/Halfwidth/Radius/Second point/Width/Undo] - pick a point or choose an option. • Draw Lines: Switches to line-drawing mode; draws line segments. • Angle: Sets the included angle of the arc segment. Prompts you: Included angle for segment (+ for ccw, - for cw) - enter a value. - type a

	value. Set end of arc or [Center/Radius] - pick point or choose an option. • Center: Sets the center of the arc segment. Prompts you: Center point - pick a point. Set end of arc or [Angle/Length of chord] - pick a point or choose an option. • Direction: Defines the direction of the tangent line in the start point of the arc. Prompts you: Direction from start - pick a point to define the direction of the tangent line. End of arc - pick a point. • Second Point - prompts you to define the second point and endpoint of a 3-point arc. See also the **Arc** command.
Close	Closes the polyline. Sets the *Closed = Yes* property of the polyline. BricsCAD automatically draws a segment from the current next-point to the starting point. 
Distance	Specifies the length and angle of the next segment; prompts you: Distance for segment - specify the length of the segment. Angle of segment - specify the angle of the segment.  Distances are measured from the last vertex. Angles are measured counterclockwise from the positive x-axis.
Follow	Draws the next segment at the same angle; move the mouse to indicate the length.
Halfwidth	Specifies the halfwidth for the current segment, as well as all segments that follow -- until you change the width or halfwidth.  You can specify different starting and ending halfwidths, which creates tapered segments; prompts you:

	Starting half-width - specify the width at the start end of the segment; enter a value, or pick two points in the drawing. Ending half-width - specify the width at the start end of the segment; enter a value, or pick two points in the drawing.
Width	Specifies the width for the current segment, as well as all segments that follow -- until you change the width or halfwidth.  You can specify different starting and ending widths, which creates tapered segments; prompts you: Starting width - specify the width at the start end of the segment; enter a value, or pick two points in the drawing. Ending width - specify the width at the start end of the segment; enter a value, or pick two points in the drawing. When you specify 0 as the width, the polyline is drawn like a line.
Undo	Undoes (undraws) the last segment.

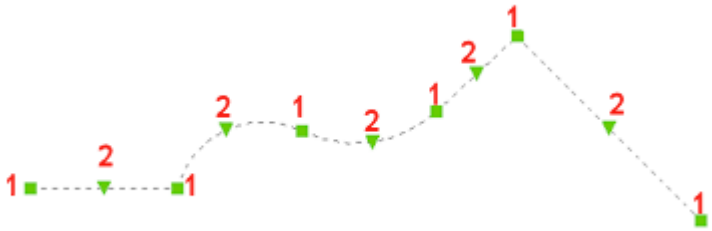
Grips Editing

Depending on the value of the GRIPS system variable polylines have grips on its endpoints and vertices and/or at the midpoint of each segment.
The value of GRIPS can be 0, 1 or 2

Value	Grips display
0	No grips
1	Endpoint grips only
2	Endpoint and midpoints grips

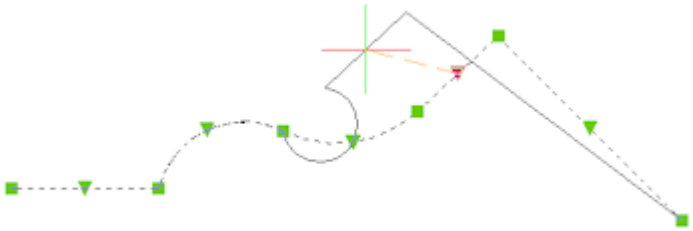
Polylines can be edited directly through grips:

1. Select the polyline. Notice that it has a square grip at each vertex (1) and a triangular grip at the midpoint of each segment (2).

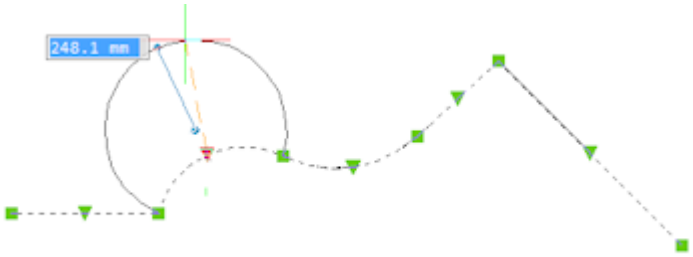


2. Do one of the following:

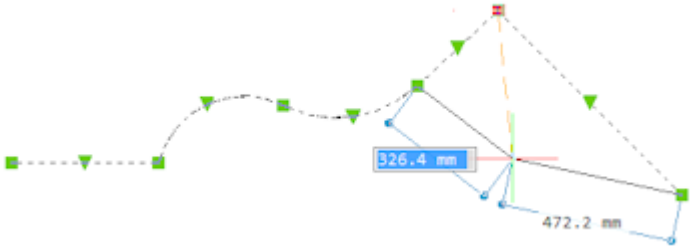
- Drag the midpoint grip of a line segment to move the segment



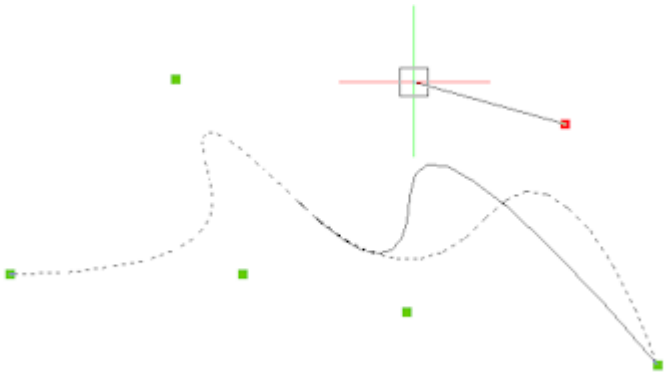
- Drag the midpoint grip of an arc segment to modify its radius.



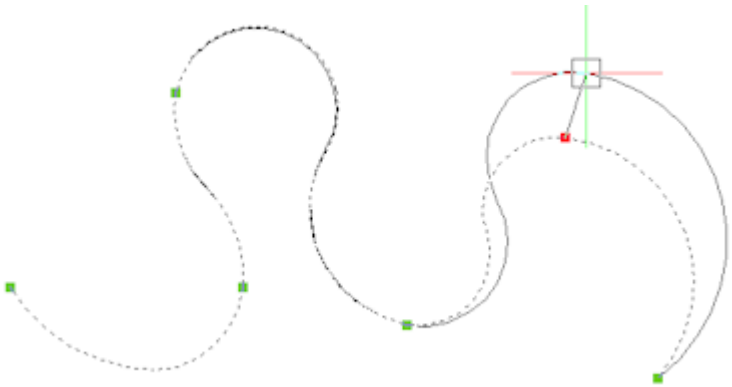
- Drag a vertex grip to move the selected vertex.



- Splined polylines have a grip at each control point, which can be stretched:



Cubic or *Quadratic* splined polyline.



Curve fit splined polyline.

Procedures

Drawing polylines
Creating boundary polylines

Related Commands

3dPoly - draws polylines with x,y,z coordinates.
Boundary - creates a polyline that forms a boundary inside a closed area through a dialog box
-Boundary - creates a polyline boundary inside closed areas through the command bar.
ConvertPoly - converts lightweight polylines to classic polylines (2D polylines) and vice versa.
Helix - creates a 2D spiral or a 3D helix.
MLine - draws multilines.
PEdit - edits polylines.
Donut - draws fat circles out of polylines.
Polygon - draws regular polygons out of polylines.
Rectang - draws rectangles out of polylines.
Solid - draws solid-filled entities with 3 and 4 sides.
Trace - draws wide lines.

Plot

Plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

Accessing the Command

command bar: *plot*

: ***plot***
Prompts you in the command bar:

Detailed plot configuration [Yes/No] <No>: (Type Y or N.)

Enter a layout name or ? <Model>: (Enter a name, type ? for a list of layout names, or press Enter to accept the default).

Enter a page setup name <>: (Enter a name, press Enter for none or type: _previous plot or <previous plot>).

Enter an output device name or ? <None>: (Enter a name, type ? for a list of device names, or press Enter to accept the default).

Write the plot to a file [Yes/No] <No>: (Type Y or N).

Save changes to model tab [Yes/No] <No>: (Type Y or N).

Proceed with Plot [Yes/No] <No>: (Type Y or N).

Detailed plot configuration:
Enter a layout name or ? <Model>: *(Enter a name, type ? for a list of layout names, or press Enter to accept the default).*

Enter an output device name or ? <dfault.pc3>: (Enter a name, type ? for a list of device names, or press Enter to accept the default).

Enter paper size or ? <Letter>: (Enter a name, type ? for a list of size names, or press Enter to accept the default).

Enter paper units [Inches/Millimeters] <Millimeters>: (Type I or M).

Enter drawing orientation [Portrait/Landscape] <Portrait>: (Type P or L).

Plot upside down [Yes/No] <No>: (Type Y or N).

Enter plot area [Display/Extents/Limits/View/Window] <Display>: (Enter an option).

Enter plot scale (Plotted millimeters = Drawing Units) or [Fit] <Fit>: (Enter a scale factor or type F).

Enter plot offset (x,y) or <0.0000,0.0000>: (Enter x,y offset distance).

Plot with plot styles [Yes/No] <Yes>: (Type Y or N).

Enter plot style table name (enter . for none) or ? <bricscad.stb>: (Enter a name, type ? for a list of style names, or press Enter to accept the default).

Plot with lineweights [Yes/No] <Yes>: (Type Y or N).

Remove hidden lines [Yes/No] <Yes>: (Type Y or N).

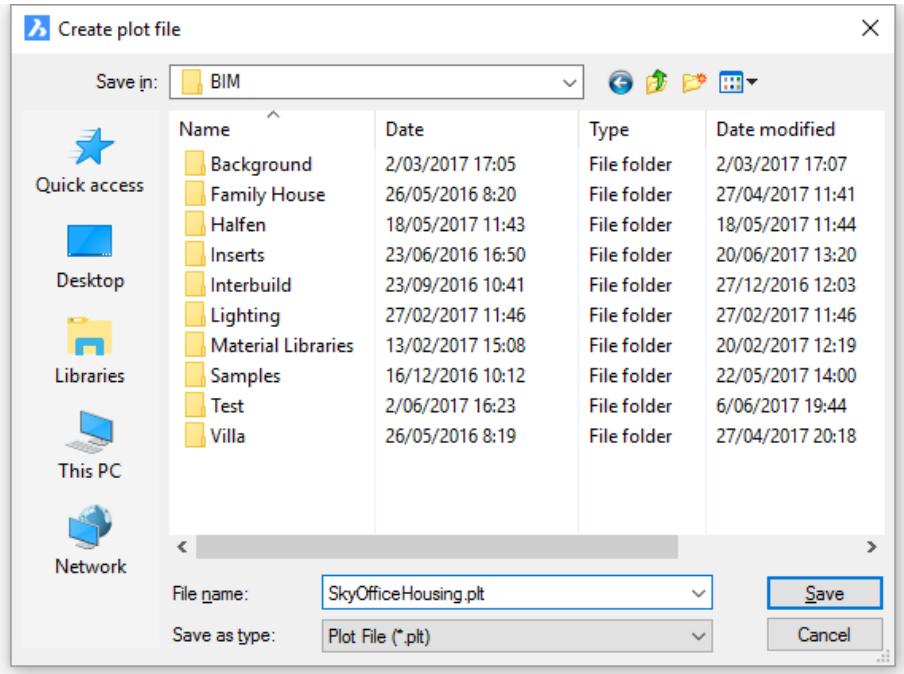
Write the plot to a file [Yes/No] <Yes>: (Type Y or N).

Save changes to model tab [Yes/No] <Yes>: (Type Y or N).

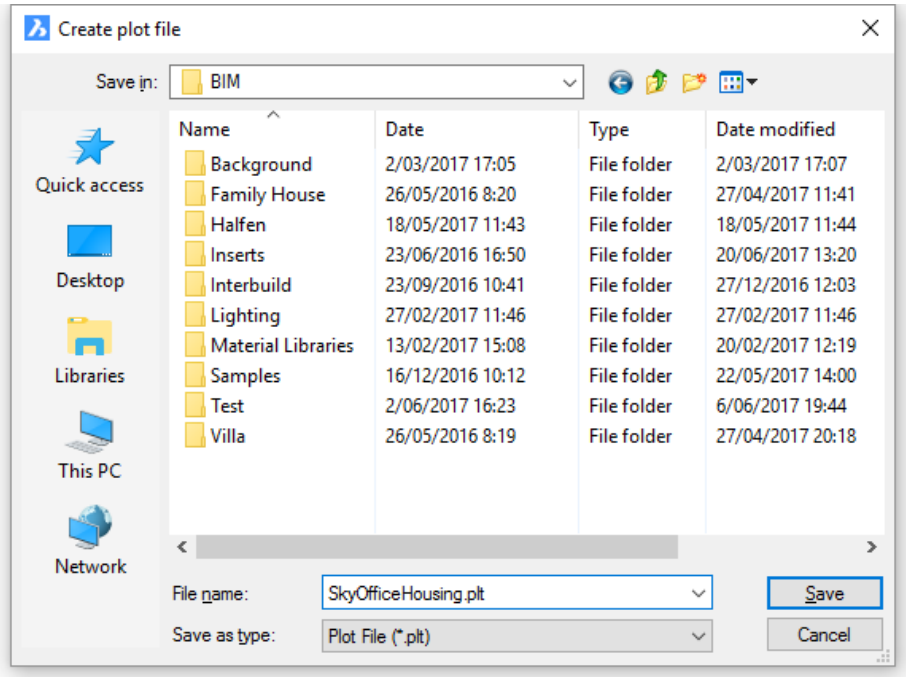
Proceed with Plot [Yes/No] <Yes>: (Type Y or N).

To configure plots with a dialog box, use **Print** command.

Command Options

Option	Description
Detailed plot configuration	Toggles detailed plot configuration: • Yes - prompts you with 16 options; see the options listed below. • No - prompts you with six options.
Enter a layout name	Specifies the name of the layout to plot. • ? - lists the names of layouts in the drawing. • Model - plots the model tab only.
Enter a page setup name	Specifies the name of a page setup. Enter a name, or press Enter for none. See PageSetup command.
Enter an output device name	Specifies the PC3 file for the output device. • ? - lists the names of devices available on this computer. When the default is None, you must specify another name.
Write the plot to a file	• Yes - plots the drawing to a PLT file. BricsCAD displays the Create Plot File dialog box.  Specify a file name, and then click Save. • No - plots the drawing to a printer or other output device.
Save changes to model tab or Save changes to **layout-name**	Toggles saving the plot parameters to the model or layout tab: • Yes - saves the plot parameters, which can be used for the next plot. • No - does not save plot parameters.
Proceed with Plot	Toggles plotting of the drawing: • Yes - plots the drawing.

	• No - does not plot the drawing; exits the command.
Detailed Configuration Option	**Description**
Enter a layout name	Specifies the name of the layout to plot. • ? - lists the names of layouts in the drawing. • Model - plots the model tab only.
Enter an output device name	Specifies the PC3 file for the output device. • ? - lists the names of devices available on this computer. When the default is None, you must specify another name.
Enter paper size	Specifies the name of the paper size: • ? - lists the names of paper sizes available for the plotter.
Enter paper units	Specifies the units used for paper sizes: • Inches - shows paper sizes in inches. • Millimeters - shows paper sizes in millimeters.
Enter drawing orientation	Determines how the drawing is plotted on the paper: • Portrait - prints the drawing so that the tall edge is upright. • Landscape - prints the drawing so that the short edge is upright.
Plot upside down	Toggles upside-down plotting: • Yes - prints drawing upside-down. • No - prints drawing normally.
Enter plot area	Specifies the area of the drawing to plot: • Display - plots the area shown in the current viewport. • Extents - plots the extents of the drawing, ensuring all non-frozen entities are plotted. • Limits - plots the limits of the drawing, as defined by the Limits command. • View - plots the current view, or a named view. See View command; prompts you: Enter view name - specify the name of a view. • Window - plots a rectangular area defined by the user; prompts you: Select first corner of window - specify one corner of the rectangle. Select second corner of window - specify the other corner.
Enter plot scale	Specifies the plot scale factor: • Plotted Inches (or Millimeters) - specifies the inches (or millimeters) on the paper. • Drawing Units - specifies the number of drawing units (inches, meters, etc). • F - fits the drawing to the paper; BricsCAD calculates the scale factor automatically.

Enter plot offset	Specifies the offset distance for the print. Enter positive or negative distances to move the drawing in the x and/or y distances: • Positive values - moves the drawing up and to the right. • Negative values - moves the drawing down and to the left. The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.
Plot with plot styles	Toggles the use of plot styles: • On - uses plot styles to determine the look of the printed drawing. • Off - does not use plot styles. Plot styles override lineweight settings.
Enter plot style table name	Specifies the plot style table to use, which assigns properties to "pens," colors, and entities. • ? - lists the names of plot styles available in the drawing. • . - specifies no plot style.
Plot with lineweights	Toggles the use the lineweights: • On - mimics lineweights assigned to the drawing in the print. • Off - ignores lineweights. This option is unavailable when Plot with Plot Styles is turned on.
Remove hidden lines	Toggles removal of hidden lines for 3D drawings: • Yes - removes hidden lines. • No - does not remove hidden lines.
Write the plot to a file	• Yes - plots the drawing to a PLT file. BricsCAD displays the Create Plot File dialog box.  Specify a file name, and then click Save. • No - plots the drawing to a printer or other output device.

Save changes to model tab or Save changes to **layout-name**	Toggles saving the plot parameters to the model or layout tab: • Yes - saves the plot parameters, which can be used for the next plot. • No - does not save plot parameters.
Proceed with Plot	Toggles plotting of the drawing: • Yes - plots the drawing. • No - does not plot the drawing; exits the command.

Related Commands

Print - plots drawings through a dialog box.
QPrint - plots drawings using the current settings.
Preview - previews drawings before they are plotted.

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.

Layer - assigns plot styles to layers.

PageSetup - assigns plot style to drawings.

ExportPdf - exports layouts in a drawing to a pdf docucment.

PlotStyle - sets the current plot style.
StylesManager - creates and attaches plot styles to drawings.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

-Publish **- prints a saved sheet list (= a list of model space or paper space layouts).**

-Plot

See Plot command.

PlotStyle

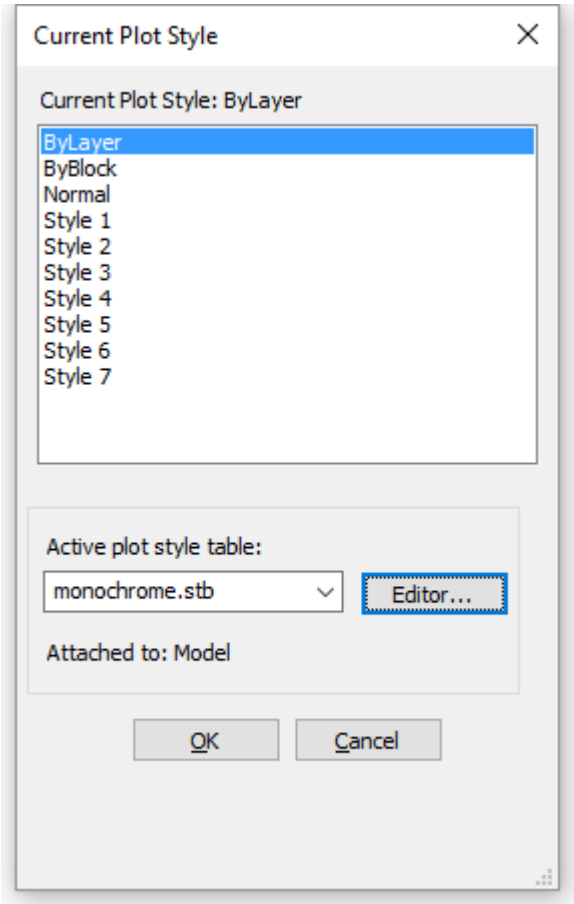
Sets the current plot style; works only when plot styles are enabled in drawings.

Accessing the Command

command bar: *plotstyle*

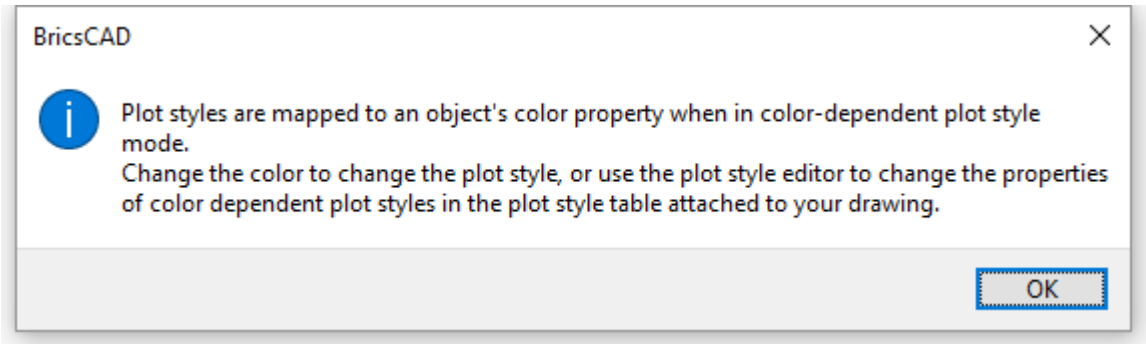
: ***plotstyle***

Displays a dialog box:



Choose a plot style name, and then click OK.

When drawings are not set up for plot styles, this command displays the dialog box:



Click OK.

Command Options

Option	Description
Active plot style table	Lists the names of STB files (short for "style table").
Editor	Displays the PlotStyle Table Editor dialog box; see the StylesManager command.

Related Commands

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.

Layer - assigns plot styles to layers.

PageSetup - assigns plot style to drawings.
Print - plots drawings with plot styles.
Properties - changes plot styles assigned to entities.
StylesManager - creates and attaches plot styles to drawings.

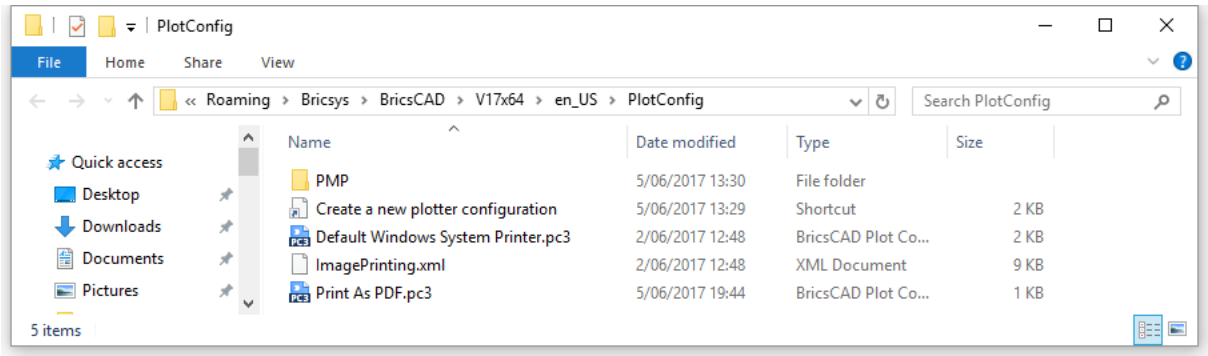
PlotterManager

Creates customized parameter PC3 files for printers and other output devices; executes the PlotConfig.exe utility program.

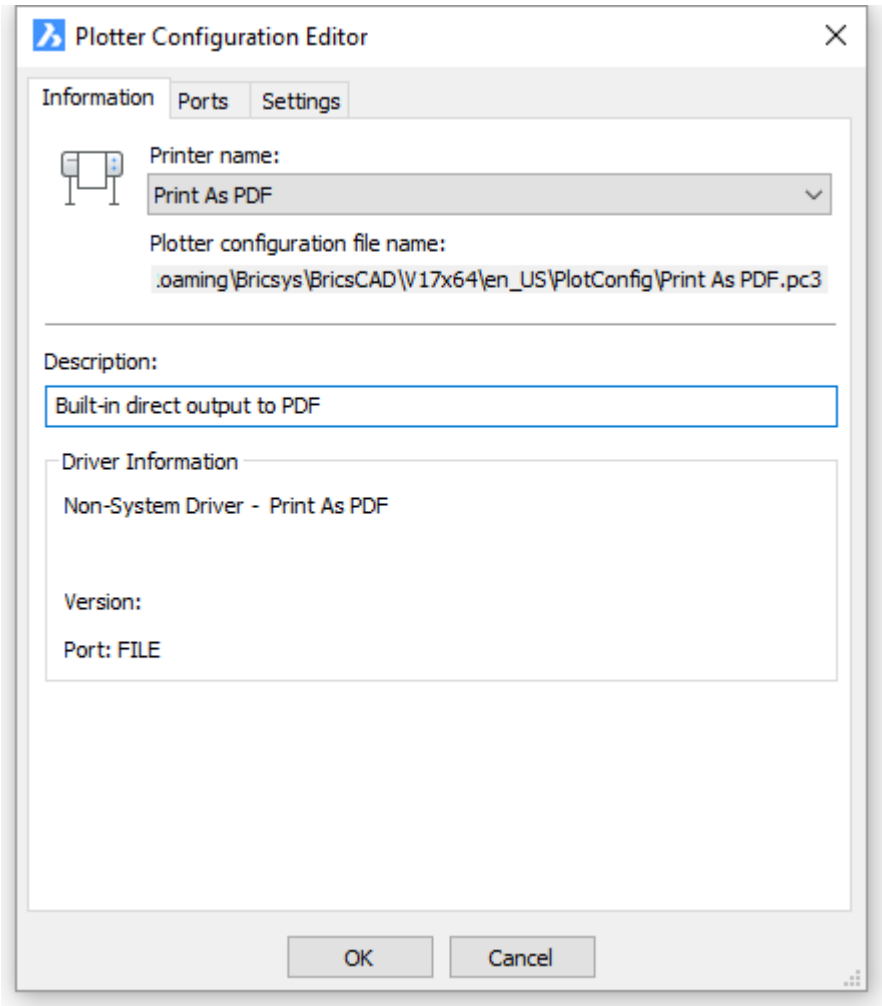
Accessing the Command

command bar: *plottermanager*
menu bar: *File | Plotter Manager*

: ***plottermanager***
Displays a window:

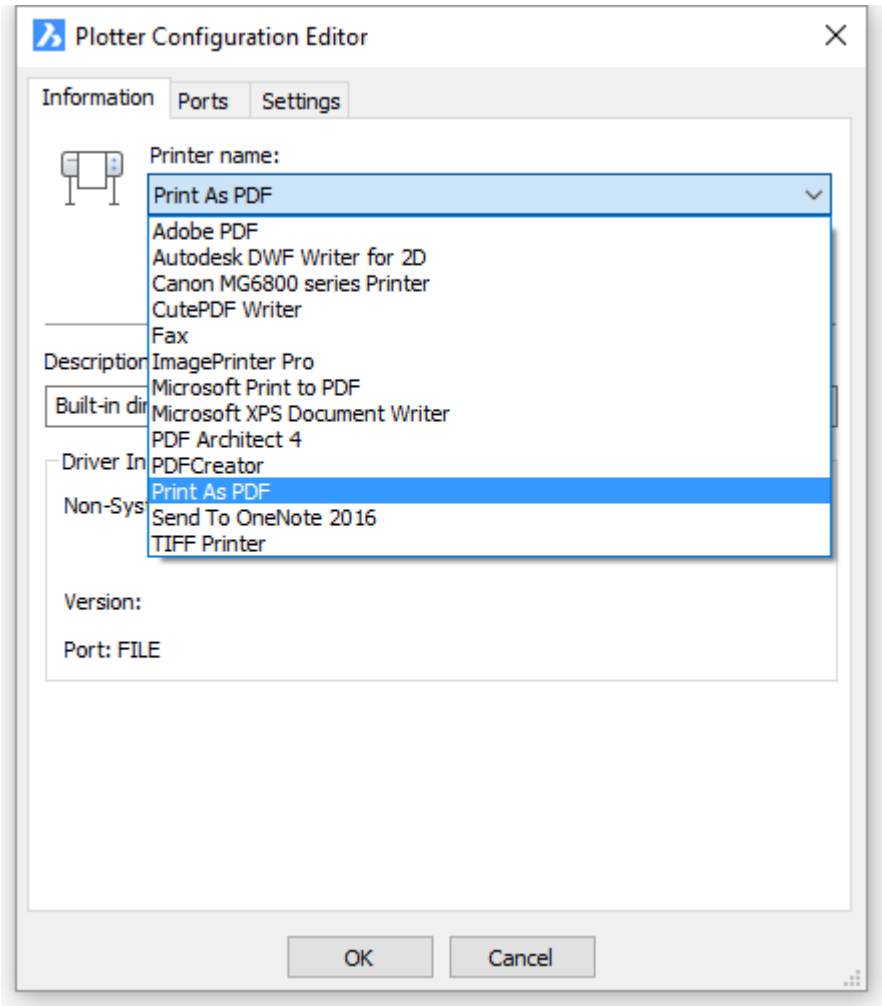


To edit an existing plotter configuration file, double click a PC3 file.



Edit the settings, and then click OK.

To create a new plotter configuration file, double-click the Create a Plotter Configuration shortcut. The Plotter Configuration Editor dialog displays:



Select a printer from the Printer name list button.

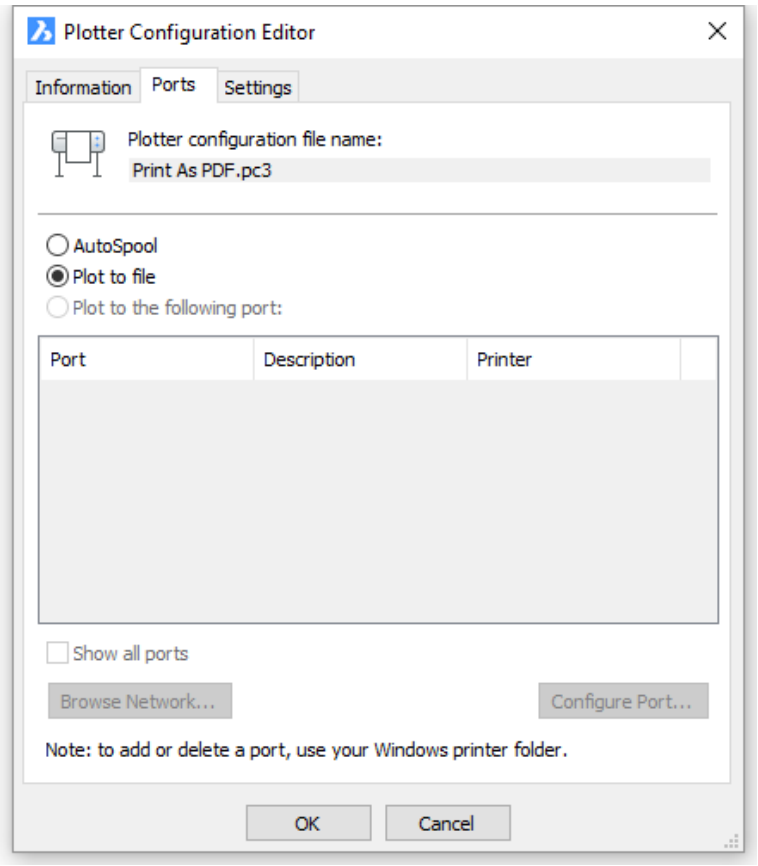
Edit the settings, ***and then click* OK.**

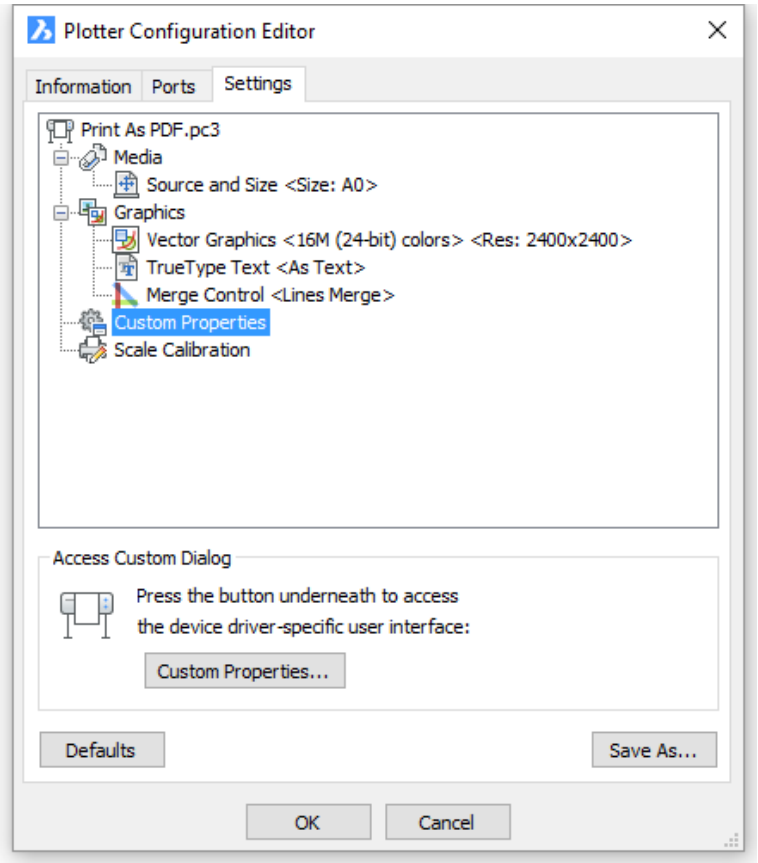
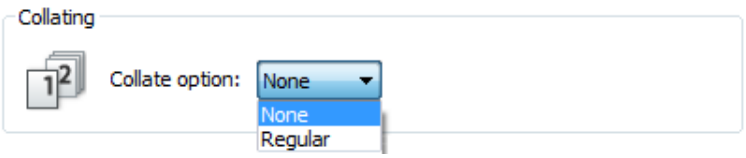
You are prompted to save the new plotter configuration file

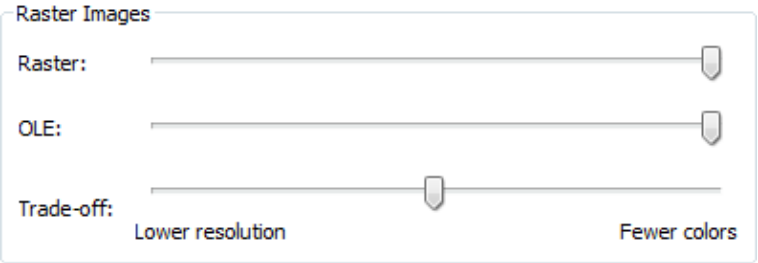
Enter a name, and then click OK.

The new PC3 file is added to the C:\Users\<loginname>\AppData\Roaming\Bricsys\BricsCAD\V...\en_US\plotconfig folder.

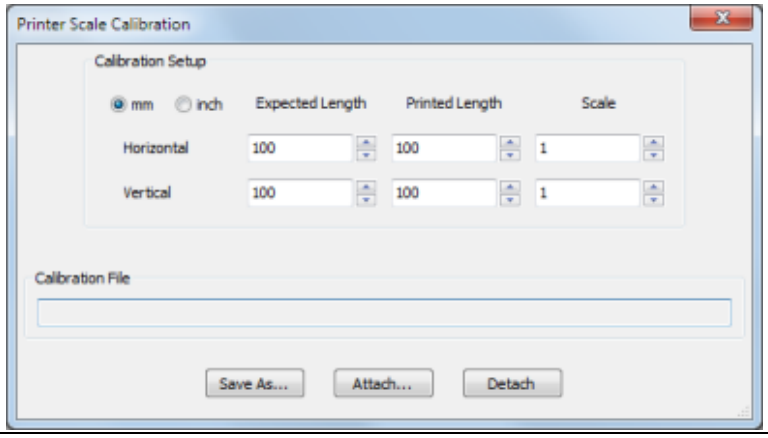
Command Options

General Option	Description
Description	Describes the plotter configuration; displayed in tooltips. This field is optional.
Ports Option  	
Plot to the following port	Plots drawings to the printer attached to the specified port.
Plot to file	Plots drawings to PLT files. These plot files can be read by other software programs.
AutoSpool	Sends drawings to spoolers, which control how and when drawings are plotted. The spooler software must be set up separately.
Show all ports	Displays the names of all local and network ports found on the current computer.

Browse Network	Opens the Browse Network window. Allows you to locate printers attached to the network.
Configure Port	Opens the Configure Port dialog box. Allows you to configure the selected port.
Settings Option  	
Media	Specifies settings for the media (paper).
 	Specifies how to collate pages: None - does not collate; drawings are printed normally. Regular - prints layouts of each drawing together. This option is useful when printing multiple copies of drawings with multiple layouts.
Graphics	Specifies how graphics are handled by the printer.

Vector Graphics (currently NOT supported)	Specifies how vector graphics are handled: Color depth - specifies the number of colors to use. Color - outputs the drawing in color. Monochrome - outputs the drawing in monochrome (black). Resolution - adjusts the resolution; lower resolution is coarser, but prints more quickly. Dithering - specifies the type of dithering, which simulates more colors at the cost of lower resolution. Vector graphics are normal BricsCAD drawings made with lines, arcs, and so on.
	Specifies how raster vectors are handled: Raster - adjusts how raster images are handled. OLE - adjust how OLE objects are printed; ranges from: • None - does not print OLE entities. • Best - prints OLE entities at highest quality. Trade-off - specifies how BricsCAD should handle drawings with very large raster images that take up much memory: • Lower Resolution - prints images with lower resolution but full colors; suitable for photographs. • Balance - prints images with as many colors and as much resolution as possible. • Fewer Colors - prints images with fewer colors but full resolution; suitable for grayscale images.

	Raster graphics are images attached to drawings.
TrueType Text TrueType: As Text As Text As Graphics	Specifies how TrueType text is handled: • TrueType as text - prints text as text; preferred when the printer holds the needed fonts. • TrueType as graphics - prints text as graphics; preferred when drawings are printed to files.
Merge Control Lines Overwrite Lines Overwrite Lines Merge	Specifies how overlapping lines are merged: • Lines Overwrite - specifies that entities with higher draw order overwrite other entities. • Lines Merge - specifies that entities are blended to merge their colors; allows entities with lower draw order to be seen.
Custom Properties Access Custom Dialog Press the button underneath to access the device driver-specific user interface: Custom Properties...	Accesses the Custom Properties dialog box. This dialog box lists properties unique to the selected output device. Its content varies according the capabilities of the device. Custom properties are not supported on the Linux platform.
User-Defined Paper Sizes & Calibration	
Plotter Calibration Access Scale Calibration Dialog Press the button underneath to manage the printer scale calibration: Scale Calibration...	Calibrates the plotter. Calibrate Plotter - displays the Calibrate Plotter wizard. See below. This option was necessary in the days of pen plotters, which could go out of alignment; today's inkjet and laser printers not to have the problem.

PMP File Name  	Specifies PMP file names. PMP is short for "plot model parameter." These files stored custom paper sizes and plotter calibration data; they are associated with PC3 plotter configuration files. Attach - displays the Open dialog box; choose a PMP file, and then click Open. Save PMP - displays the Save As dialog box; name the file, and then click Save. Detach - removes the attached PMP file.
Import	Imports PCP and PC2 files; displays the Plotting Components dialog box. 1. Click OK. 2. Displays the Import dialog box. 3. Choose a PCP or PC2 file, and then click Import. PCP and PC2 are older formats of the plotter configuration files.
Save As	Displays the Save As dialog box; saves settings as PC3 files. 1. Enter a file name, and then click Save. PC3 is short for "plotter configuration version 3"; these files store parameters for specific plotters. You can have multiple PC3 files for each plotter or printer.
Defaults	Resets all options to their default values; prompts you: Do you want to set everything to defaults? **- type Y or N.**

Procedures

To create a plotter configuration file
To edit a plotter configuration
To assign a plotter configuration

Related Commands

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.

PageSetup - assigns plot style to drawings.

PlotStyle - sets the current plot style.
Print and Plot - plot drawings.
StylesManager - creates and attaches plot styles to drawings.

Point

Draws point entities.

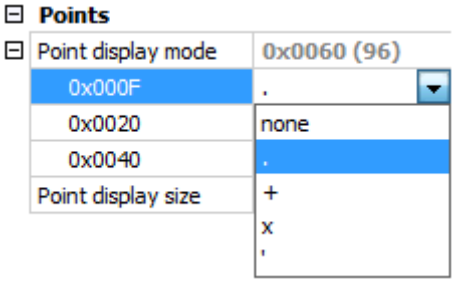
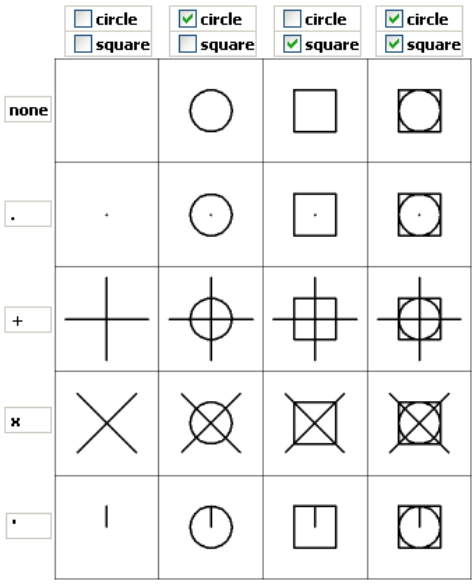
Accessing the Command

command bar: *point*
alias: *po*
menu bar: *Draw | Point*
toolbar: *Draw |*

: ***point***
Prompts you in the command bar:

Location of point or [point Settings.../Multiple points]: (Pick a point, enter x,y,z coordinates, or enter an option.)

Command Options

Option	Description
Location of point	Specifies the location of the point in the drawing. Pick a point, or enter x,y,z coordinates.
Point settings	Displays the **Points** section of the Settings dialog box:  **Point display mode** - specifies the look of points. 

	Point display size - specifies the size of points.
Multiple points	Repeats the command to draw multiple points; keeps prompting you: **Location of point or [point Settings...]** - specify a point or press Enter to stop.

Procedures

Defining the point display settings
Drawing points

Grips Editing

Points can be edited directly through grips:

1. Select the point. Notice that it has one grip.
2. Drag the grip to move the point.

Related Commands

DdPType - defines the size and shape of points.
Divide - places evenly-spaced points along entities.
Measure - places equi-distant points along entities.

Pointlight

Creates a point light.

Accessing the Command

command bar: *pointlight*
menu bar: *View | Rendering | Lights | New point light*
toolbar: *Lights |*

: ***pointlight***
Prompts you in the command bar:

Specify source position <0,0,0>: *(specify the position of the light.)*

If the LIGHTINGUNITS system variable is 0 (No lighting units), the following prompt displays:

Enter an option to change [Name/Intensity/Status/shadoW/Attenuation/Color/eXit] <eXit>:

If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), the following prompt displays:

Enter an option to change [Name/Intensity factor/Status/Photometry/shadoW/Attenuation/filterColor/eXit] <eXit>:

Command Options

Option	Description
Name	Enter light name <Pointlight1>: (Enter a name or press Enter to accept the default name.)
Intensity Factor Intensity	Enter intensity (0.00 - max float) <1.0>: (Enter the intensity. Values between 0.00 and the maximum value supported by your system are accepted.)
Status	Enter status [ON/OFf] <ON>: (Choose ON to include the light when calculating a rendering.)

Photometry	**Enter a photometric option to change [Intensity/Color/eXit] <Intensity>:**
	• Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>: (Enter the intensity expressed in Candela (Cd).)
	  • Enter Flux (Lm) : (Enter the luminous flux expressed in Lumen (Lm).)
	  • Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>: (Enter the illuminance expressed in Lux (Lx) or Foot-candle (Fc).)
	  • Enter Distance <1.0>: (Enter a distance expressed in drawing units.)
	• Enter color name or enter an option [?/Kelvin] <D65>:
	  • Enter color name(s) to list <*>: (Enter a color name from the list. Type * to see the list)
	  • D65
	  • Fluorescent
	  • Coolwhite
	  • Whitefluorescent

	• daylightfluorescenT • Incandescent • Xenon • Halogen • Quartz • Metalhalide • mErcury • Phosphormercury • highpressureSodium • Lowpressuresodium • Enter Kelvin temperature <3600.0>: (Enter a color temperature expressed in degrees Kelvin (K).) Candela: The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela. Lumen: The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens. Illuminance: In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human brightness perception. If LIGHTINGUNITS=2 (International lighting units): In SI derived units illuminance is measured in lux (lx) or lumens per square metre. If LIGHTINGUNITS=1 (American lighting units): The foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux, although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux. Kelvin temperature: Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K).

	Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700&endash;3,000 K) are called warm colors (yellowish white through red).

Shadow	Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>: (Enter a shadow setting.)
	• Off: Disables the calculation of shadows for the light.
	• Sharp: Displays shadows with sharp edges. Use this option to increase performance.
	• Soft mapped: Displays realistic shadows with soft edges.
	Prompts you: Enter map size [64/128/256/512/1024/2048/4096] <256>:
	• Soft sampled: Displays realistic shadows with softer shadows based on extended light sources.
	Prompts you: Enter an option to change Shape/sAmples/Visible/<eXit>:
	  • Enter shape [Disk/Rect] <Disk>:
	  • Enter radius <0.0>: (Enter the radius of the disk.)
	  • Enter Length <0.0>: (Enter length of the rectangle.)
	  • Enter Width <0.0>: (Enter width of the rectangle.)
	  • Enter Shadow Sample <16>:
	  • Enter Shape Visibility [Yes/No] <No>:

Attenuation	**Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit] <eXit>:**
	Enter an option or press Enter to return to the base options.
	• Enter attenuation type [None/Inverse linear/inverse Squared] <None>:
	  • None: No attenuation. The distance to the light source has no influence.
	  • Inverse linear: The attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.
	  • Inverse squared: The attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.
	• Specify start limit offset <1.0>: Define the point where the light starts to shine, measured from the center of the light.
	• Specify end limit offset <10.0>: Define the point where the light stops to shine, measured from the center of the light.

Filter Color Color	Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>: Defines the color of the light. • True Color (RGB): The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • Index color: Specifies an AutoCAD Color Index color. • HSL: Specifies the color using three parameters Hue, Saturation and Lightness.
Exit	Concludes the command and creates the light.

Related Commands

Distantlight - creates a distant light.
Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Render - calculates a photorealistic image of the 3D model.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
Weblight - creates a web light.

Polygon

Draws regular polygons, from 3 to 1024 sides.

Accessing the Command

command bar: *polygon*
alias: *pol*
menu bar: *Draw | Polygon*
toolbar: *Draw | Polygon |*

: ***polygon***
Prompts you in the command bar:

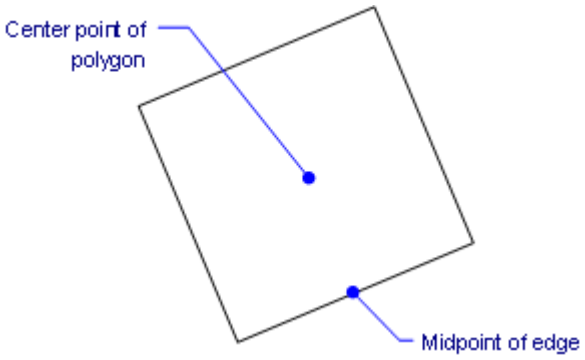
Polygon: Set number of sides or [Width of line/Multiple polygons] <4>: **(Specify the number of sides, or enter an option.)**

Specify by: Set center of polygon or [specify by Edge]: **(Pick the center point, or type E.)**

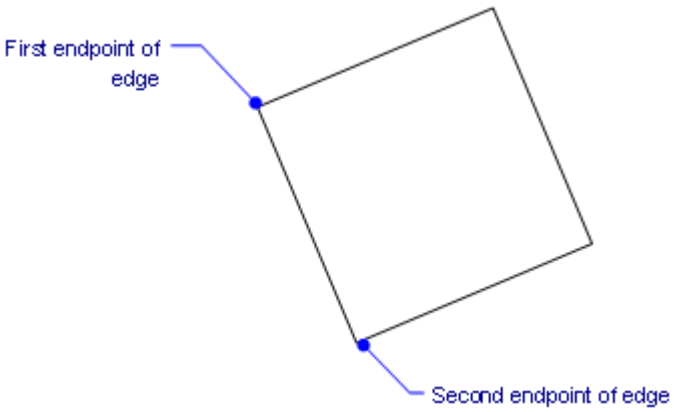
Specify by: Select midpoint of side or [specify by Vertex]: **(Pick a point, or type V.)**

This command draws polygons by three methods:

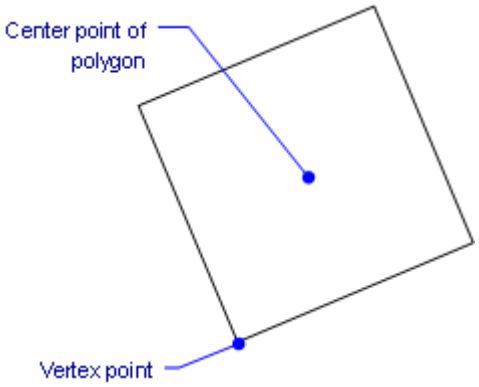
- Location of center point and the midpoint of a side:



- Length of one edge:

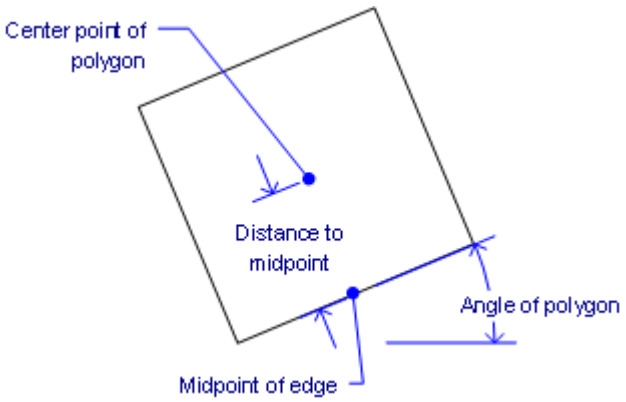


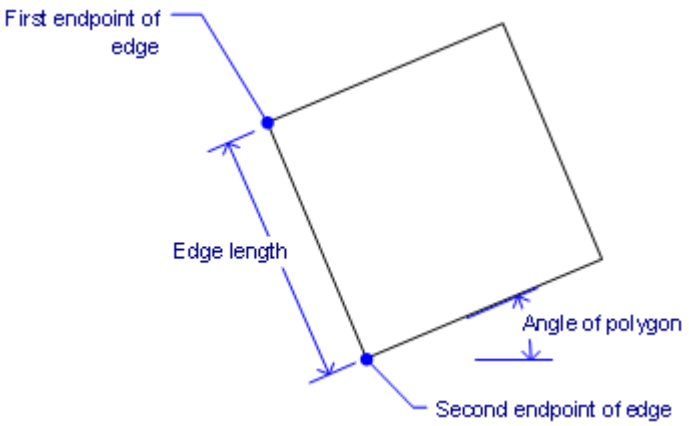
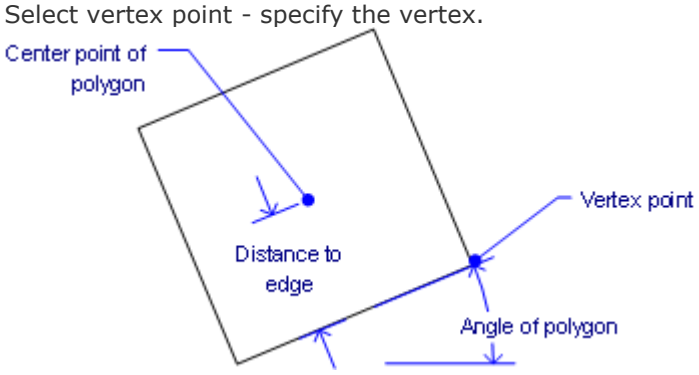
- Location of center point and one vertex:



This command draws regular polygons, where each side is the same length.
To draw irregular polygons, use the **R**ectang or **PLine** commands.

Command Options

Option	Description
Number of sides	Specifies the number of sides; enter a number between 3 and 1024.
Multiple polygons	Repeats the command to draw additional polygons.
Width of line	Specifies the width of the edge lines. Polygons are made of polylines.
Center of polygon	Specifies the center point of the polygon.
Midpoint of side	Specifies the midpoint of one of the sides.  The midpoint of the side determines the size and angle of the polygon.
Edge	Specifies the length of an edge; prompts you: First endpoint of edge - specify the edge's start point. Second endpoint of edge - specify the edge's endpoint.

	 The length of the side determines the size and angle of the polygon.
Vertex	Specifies a vertex (corner of the polygon); prompts you: Select vertex point - specify the vertex.  The location of the vertex determines the size and angle of the polygon.

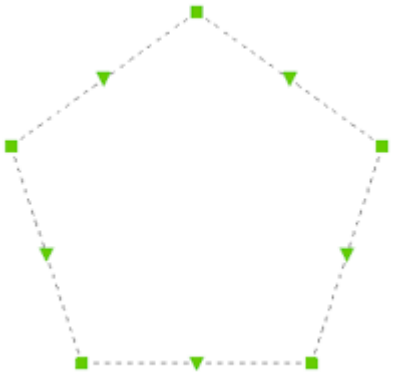
Procedures

Drawing polygons

Grips Editing

Polygons can be edited directly through grips:

1. Select the polygon. Notice that it has a square grip at each vertex and a triangular grip at the midpoint of each segment.



2. Drag a square grip to move a vertex.

3. Drag a triangular grip to move a segment.

Notice that the polygon is no longer regular.

Related Commands

Rectang - draws rectangles and squares.
PEdit - edits polygons.

PolySolid

Creates wall-like or pitched roof 3D solids.

Accessing the Command

command bar: *polysolid*
alias: *pso*
menu bar: *Model | 3D Solids | Polysolid*
Quad: *Model*
toolbar: *Solids |*

: ***polysolid***
Prompts you in the command bar:

Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>: (Specify a point (1), right click to select an existing entity or enter an option.)

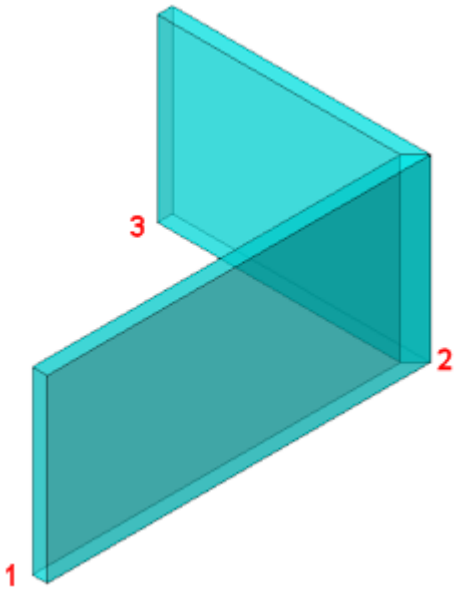
Set next point or [draw Arcs/Distance]: (Specify a point (2), enter an option or press Enter to exit.)

Set next point or [draw Arcs/Distance/Follow/Undo]: (Specify a point (3), enter an option or press Enter to exit.)

Set next point or [draw Arcs/Close/Distance/Follow/Undo]: (Specify a point, enter an option or press Enter to exit.)

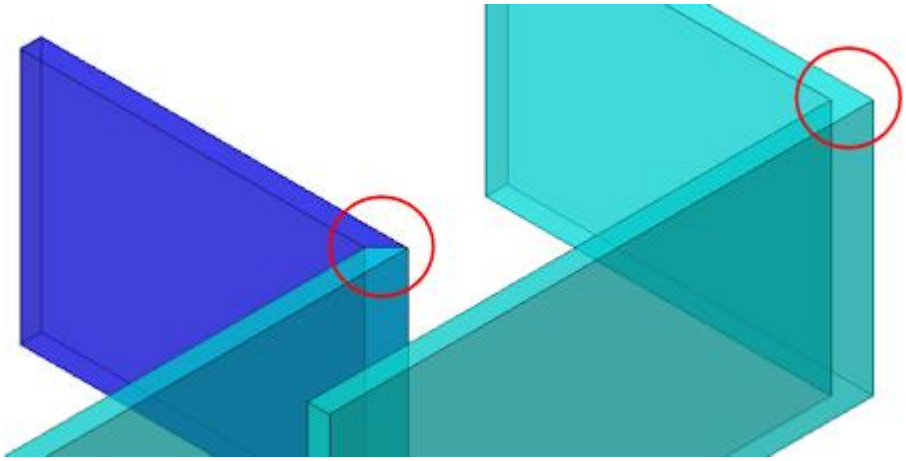
If Dynamic Height is ON, prompts you:

Height of polysolid <xxx.xx>: (Press Enter to accept the default height or type a value.)

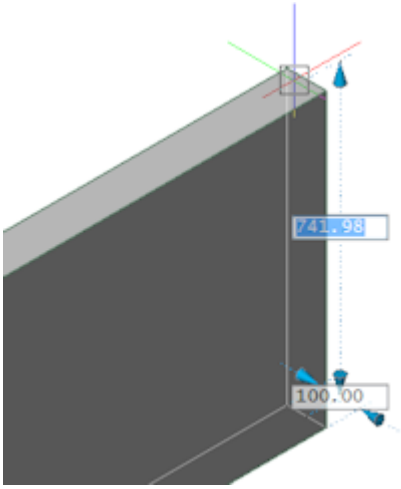


The height of the polysolid is created perpendicular to the XY-plane of the current coordinate system (WCS or UCS) or dynamic UCS.

Command Options

Option	Description
Height	Sets the height of the polysolid. The default height is saved to the PSOLHEIGHT system variable. The default height is used if the Dynamic option is OFF.
Width	Sets the width of the polysolid. The default width is saved to the PSOLWIDTH system variable. If dynamic dimensions are active,
Justification	Defines the position of the polysolid entity with respect to the base line. Prompts you: **Select justification [Left/Right/Center] <Center>:** Choose an option. Justification is *Center* by default. This option is not saved and will be reset to default when BricsCAD is closed. Dynamic justification: If Dynamic is ON: when the command prompts for the height of the polysolid, press the Ctrl key to toggle the justification between Left, Right and Center. If the HOTKEYTIP system variable is ON, the *Hot Key Tip* assistant indicates the currently selected option.
Separate Solids	Defines whether a chain of L-bisector* connected solids are created (left) or a single multi-segment solid is created (right). *Separate Solids* is ON by default. This option is not saved and will be reset to default when BricsCAD is closed. *Bisector connection lines are removed in sections depending on the Union Section property of the material(s) in the composition that is assigned to the connected solids.  Prompts you: Separate solids On/OFF <On>: • ON: Creates separate solids

	• OFF: Creates a single multi-segment solid.

Dynamic	Defines whether the height of the polysolid is fixed or not.
	Prompts you:
	Dynamic height On/OFF <On>:
	• ON: you will be prompted to define the height of the polysolid entity:
	Height of polysolid <xxx.xx>:
	Do one of the following:
	  • Press Enter to accept the default height.
	  • Type a value in the command bar
	  • Click a point in the drawing.
	• OFF: the height of the polysolid is defined by the PSOLHEIGHT system variable. The width is defined by the PSOLWIDTH system variable. Both variables save the most recently used width and height for polysolids.
	If set ON, and Dynamic Dimensions is ON, you can define the width and height of the polysolid entity in the dynamic entry fields. Press the TAB key to switch to the other field. Press Enter to confirm the current values.
	 

Entity	Prompts you to select an existing linear entity: **Select polysolid base:** Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs and splines are accepted.
Arc	Starts drawing arc segments. Prompts you: **Set end of arc or [draw Lines/CLose/Direction/Second point/Undo]:** Enter an option: • End of arc: Enter a point. The arc is tangent to the previous segment. • Draw Lines: Starts creating linear segments. • Direction: Lets you define the direction of the tangent at the start point of the new arc segment. • Secondpt: Prompts you to specify a point along the arc's circumference.

Distance	Lets you define the length and angle of a linear segment. Prompts you: Distance for segment: Type a value. Angle of segment: Type a value. If Dynamic Dimensions is ON the length and angle of the segment can by defined in the dynamic entry fields.
Follow	When switching from arc segments to linear segments, creates the linear segment tangent to the arc segment.
Close	Creates a closing segment between the previous point and the start point of the polysolid. If you snap to the startpoint of the currently created chain, the chain is closed automatically. This option is available after creating two polysolid segments.
Undo	Removes the previous segment.

Procedures

Creating a single wall
Creating a chain of L-connected walls

Related Commands

BimConnect - Creates an L-connection between minor faces of two solids.
BimDrag - drags major faces of solids while preserving connections with minor faces. Drags minor faces; optionally connects minor faces to major faces of other solids.

Preview

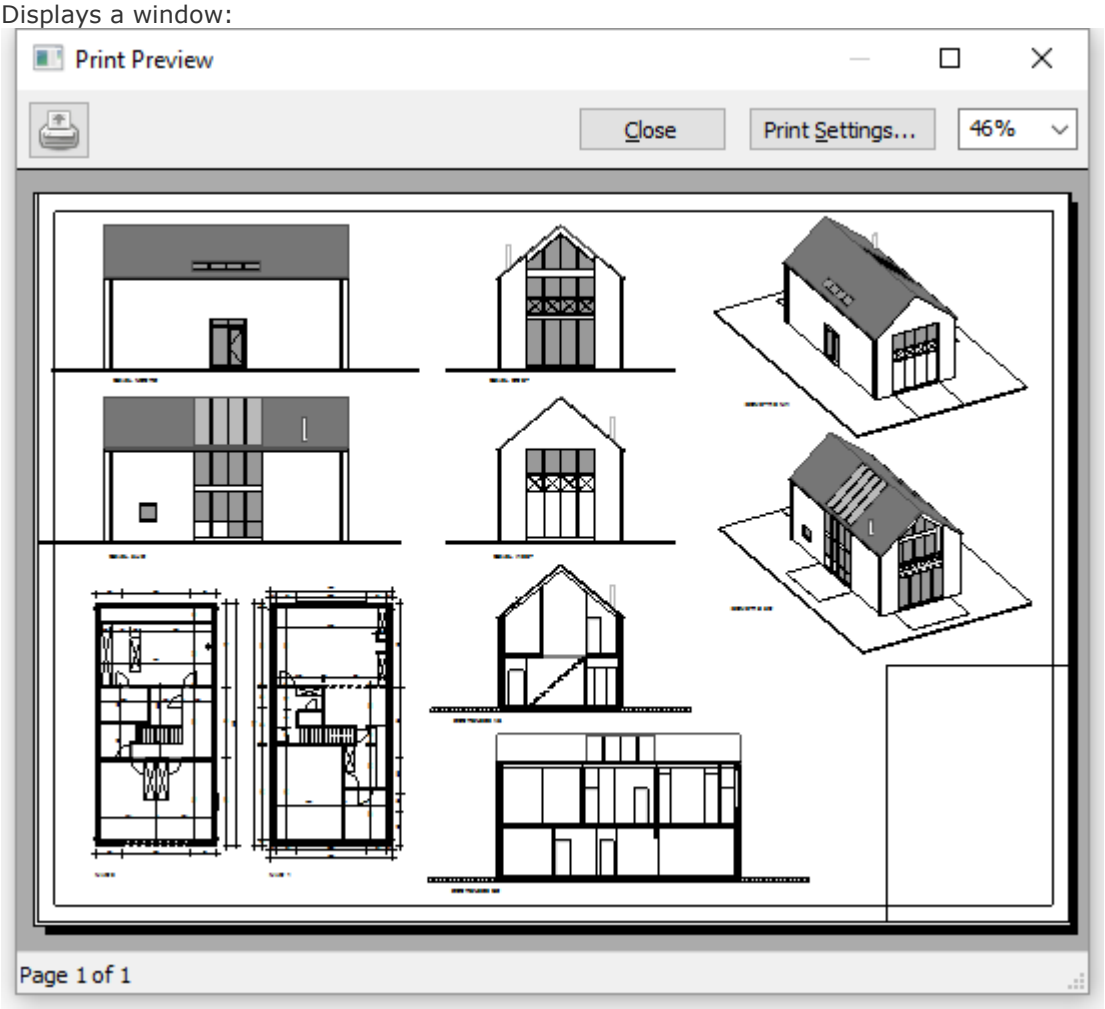
Displays the Preview window for checking drawings before plotting.

Accessing the Command

command bar: *preview*
aliases: *ppreview, pre*
menu bar: *File | Print Preview*
toolbar: *Standard |*

: ***preview***

Displays a window:



Click Print to plot the drawing, or click Close to return to BricsCAD.

Use the mouse scroll wheel to zoom.

Use the scroll bars to pan.

Command Options

Option	Description
Print Settings	Displays the Print Settings dialog box. See PageSetup command.
Print	Prints the drawing; does not display the Print dialog box. See Print command.

Close	Closes the preview window and returns to the BricsCAD drawing window.

Related Commands

Print - prints drawings through a dialog box.
Plot - plots drawings through the command bar.

Print

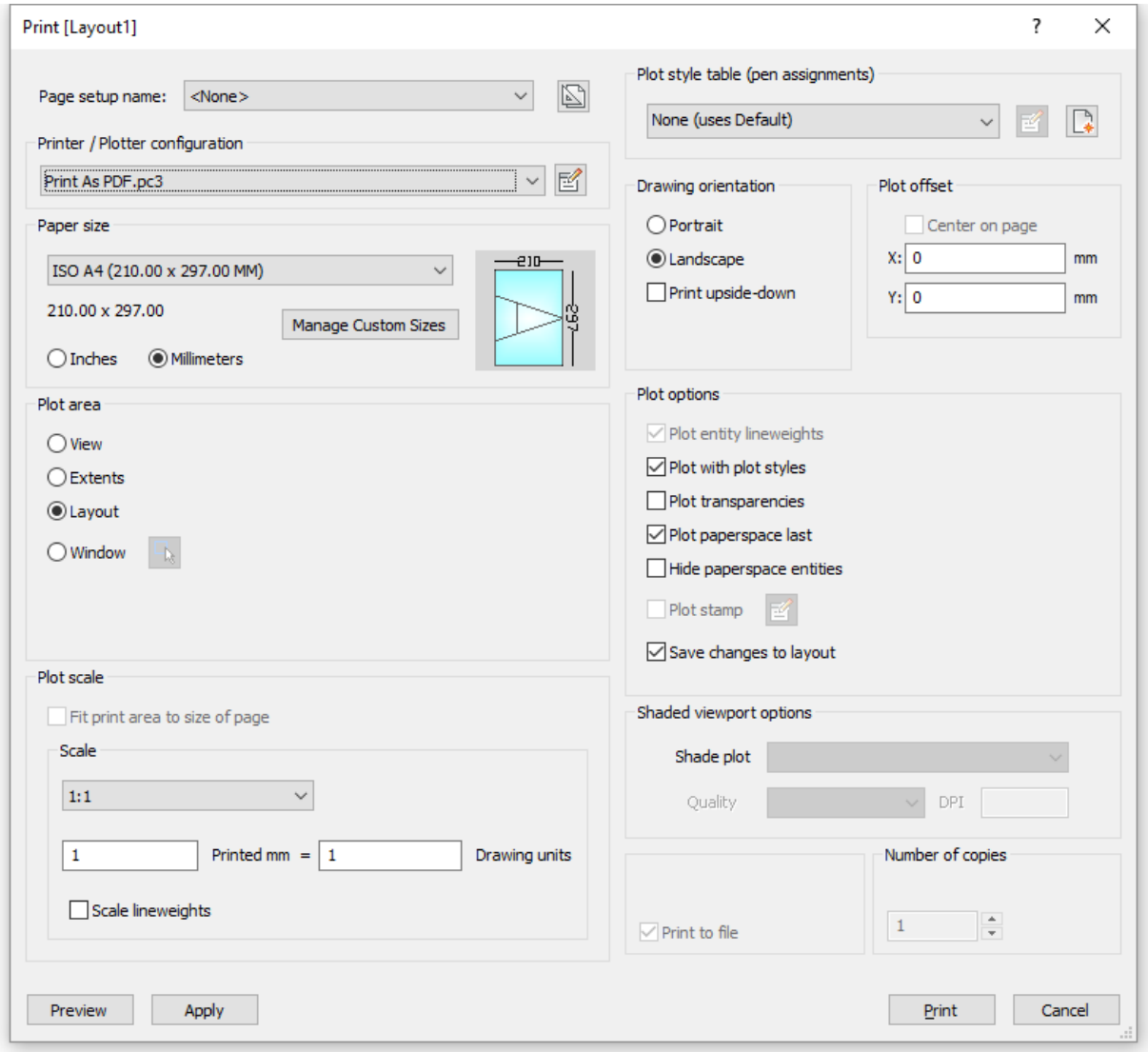
Prints drawings to plotters and files through a dialog box.

Accessing the Command

command bar: *print*
shortcut: *Ctrl+P*
menu bar: *File | Print*

: ***print***

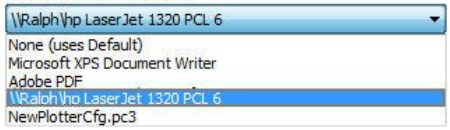
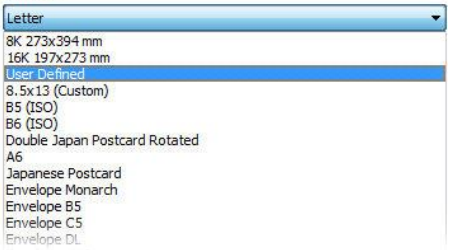
Displays a dialog box:

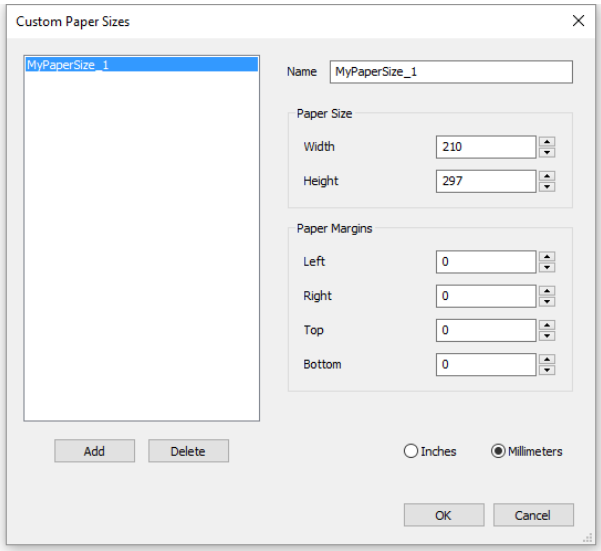


Choose options, and then:
click Preview to preview the plot on simulated paper.
click Apply to apply the changes.
click Print to print the drawing.

click Cancel to close the Print dialog box.

Command Options

Option	Description
Page setup name: 	Recalls plot settings used previously: • None - uses options saved in the current layout or model space. • Previous Plot - uses options that were saved from the last time this dialog box was used. • **Page Setup** - lists the named page setups in the drawing. Click the *Save current settings to page setup* button ( ) to create a new page setup (see the PageSetup command).
Printer/Plotter Configuration	
Printer / Plotter configuration  	Selects the printer or plotter. BricsCAD works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices. In addition, BricsCAD uses predefined printer parameters stored in PC3 files. See **PlotterManager** command.
Edit plotter configuration 	Opens the printer's Properties dialog box for customizing the printer's parameters.
Paper Size	
Paper size  	Chooses the size of paper. You can select standard sizes listed by the droplist; these are sizes supported by the printer. While printer may appear to support many different sizes, you should select only the size of paper that is actually in the printer. If User Defined appears in the list, the paper's dimensions are specified by the printer's Properties dialog box. The image indicates the size, position and orientation of the current plot area on the selected paper size.
Units 8.50 x 11.00 Inches Millimeters	Switches paper measurements between imperial and metric. 215.90 x 279.40 Inches Millimeters
Manage Custom Sizes	Allows to manage custom paper sizes when *Print as PDF.pc3* is selected in the Printer/Plotter Configuration list. Displays a dialog box:

	
Plot Area	
Plot area View Extents Limits Window	Specifies which area of the drawing to print: • View - prints the current view or a named view. See View command. Choose a view from the droplist: • Extents - prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not taken into account when calculating the extents. • Limits - prints the limits of the drawing, as specified by the Limits command. • Window - prints a rectangular area of the drawing. You define the rectangular area by entering x,y coordinates or by clicking the Select Area to be Printed button.
Select area to be printed 	Specifies the rectangular area to be printed. Click the button; the dialog box is dismissed temporarily, and you are prompted: Select first corner of window - specify one corner of the rectangle. Select second corner of window - specify the other corner. You can pick points or enter x,y coordinates. The Print dialog box returns, with the pair of x,y coordinates added to the Windows Plot Area boxes.

Windowed plot area X: 4.777 Y: 10.949 X: 9.735 Y: 15.907	Specifies the x,y coordinates of the lower-left and upper-right corners of the rectangular area to be printed. You can enter x,y coordinates, or click the button to pick points in the drawing.
Plot Scale	
Plot scale ☑ Fit print area to size of page	Scales the drawing so that it fits the printable area of the paper. The scale is calculated automatically by BricsCAD, and takes into account the print area of the drawing. The printable area is the size of the paper, minus the margins. The margins are the strips along the four edges that the printer uses for handling the paper. When this option is turned on, you cannot specify the scale factor.
Scale 1:1 Custom 1:1 1:2 1:4 1:8 1:10	Specifies the scale factor of the printed drawing. Choose a scale factor from the droplist, or select *Custom* to specify a scale factor. Available scales in the list can be edited using the ScaleListEdit command. For scale factors like 1:5, the drawing is scaled smaller. For scale factors like 5:1, the drawing is scaled larger.
1 Printed inches = 1 Drawing units	When printing Model Space page setups: Specifies user-defined scale factors. For example, to print a drawing of 50'-long house on paper 11" wide, enter: Printed Inches = 11" Drawing Units = 50' BricsCAD works out the scale factor automatically. When printing a Paper Space layout: The Scale setting specifies the relation between paper units and drawing units. Paper units are either inches or millimeters. Drawing units can be feet, centimeters, meters, ... Scale setting when drawing units are feet: 12 Printed inches = 1 Drawing units Scale setting when drawing units are cm: 10 Printed mm = 1 Drawing units
Scale lineweights	If on, lineweights are scaled with respect to the plot scale.

Plot Style Table	
Plot style table (pen assignments) None (uses Default) None (uses Default) Bricscad.ctb Default.ctb MonoChrome.ctb	Specifies the plot style table to use, which assigns properties to "pens," colors, and entities. • If the drawing does not use plot styles, then only CTB (color-based table) files are listed. • If the drawing uses plot styles, then only STB (style-based table) files are listed. Plot style table (pen assignments) Default.stb None (uses Default) Bricscad.stb Default.stb MonoChrome.stb When you switch from None to a named plot style, BricsCAD prompts you: Assign plot style table to all layouts? - click Yes or No. • Yes - assigns the CTB or STB plot style file to all layouts. • No - assigns the plot style file to the current layout only.
Edit Plot Style	Opens the Plot Style Table editor dialog box. See **PlotStyle** command. This button is available only when the drawing uses STB plot styles.
Create New Plot Style	Creates new plot styles; runs the Add Plot Style Table wizard. • If 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). See **StylesManager** command.
Display plot styles	Toggles the display of plot style names.

Paper Orientation	
Drawing orientation Portrait Landscape Print upside-down	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.
Plot Offset	
Plot offset X: 0 mm Y: 0 mm	Specifies the offset distance for the print. Enter positive or negative distances to move the drawing in the x and/or y distances: • Positive values - moves the drawing up and to the right. • Negative values - moves the drawing down and to the left. The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.
Center on page	Centers the print on the page. The center of the drawing is positioned in the center of the paper. Together with Fit Print Area to Size of Page, this option is excellent for prints where scale is unimportant.
Plot Options	
Plot entity lineweights	Toggles the use the lineweights: • On - mimics lineweights assigned to the drawing in the print. • Off - ignores lineweights. This option is unavailable when Plot with Plot Styles is turned on.
Plot with plot styles	Toggles the use of plot styles: • On - uses plot styles to determine the look of the printed drawing. • Off - does not use plot styles.

	Plot styles override lineweight settings.
☑ Plot transparencies	Plots entities using the value of the Transparency property.
☑ Plot paperspace last	Toggles printing order: • On - prints model space entities, followed by paper space entities. • Off - prints paper space entities first, followed by entities in model space. This option is available only when printing layouts; it is unavailable in model tab.
☑ Save changes to layout	Toggles saving of print options: • On - saves changes made to this dialog box with the layout. • Off - does not save changes. The next time you use this dialog box, you can select "Layout" from the Use Plot Settings From droplist.
☑ Plot stamp	Toggles use of the plot stamp: • On - applies plot stamp data to the print. • Off - does not apply plot stamps. Click the Edit Plot Stamp button to change plot stamp data.
	Displays the Plot Stamp dialog box.
☑ Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
Shaded Viewport Options	
Shaded viewport options Shade plot: As displayed Quality: DPI	Allows to override the visual style of the current view when printing model space. 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).
☑ Print to file	If checked, redirects the print output to a *.plt file.
Number of copies: 1	Specifies the number of copies to print. Enter a number or click the buttons to change the value.

Preview	Displays a preview of the print. In the preview window, click Print Settings button to return to this dialog box. See **Preview** command.
Apply	Applies the changes. You can click Cancel to exit the dialog box. The changes are remembered the next time you use this command. This is like using the **PageSetup** command.
Print	Prints the drawing.
Cancel	Returns to the drawing editor.

Procedures

General procedure for printing
To open the Print dialog
To choose the paper size
To choose the paper units
To choose the drawing orientation
To define the plot offset
To specify the area to plot
To define the plot scale
To set the plot options
To define the plot stamp
To specify the number of copies
To see a preview of the plot output

Related Commands

Plot - plots drawings through the command bar.
QPrint - plots drawings using the current settings.
Preview - previews drawings before they are plotted.

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.

ExportPdf - exports layouts in a drawing to a pdf docucment.

Layer - assigns plot styles to layers.

PageSetup - assigns plot style to drawings.

PlotStyle - sets the current plot style.
Properties - changes plot styles assigned to entities.
StylesManager - creates and attaches plot styles to drawings.
ScaleListEdit - edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

-Publish - prints a saved sheet list (= a list of model space or paper space layouts).

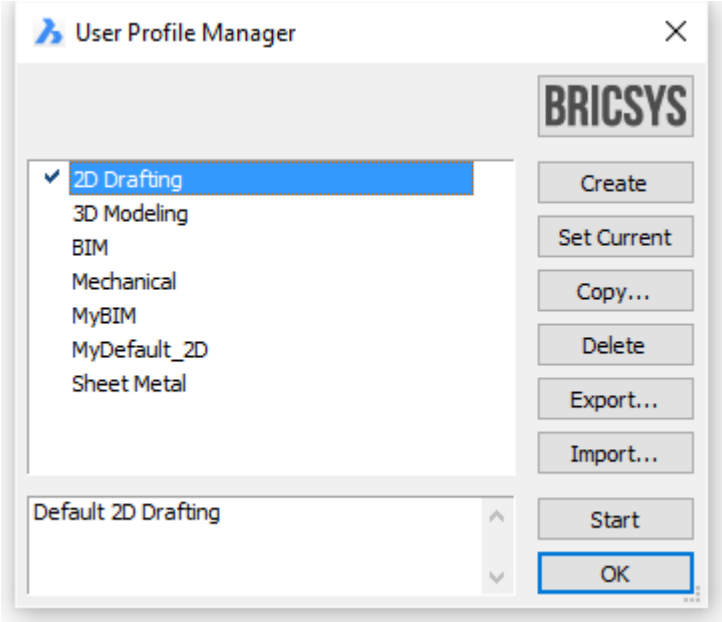
ProfileManager

Allows to set current, create, copy, delete, import and export user profiles through a dialog box.

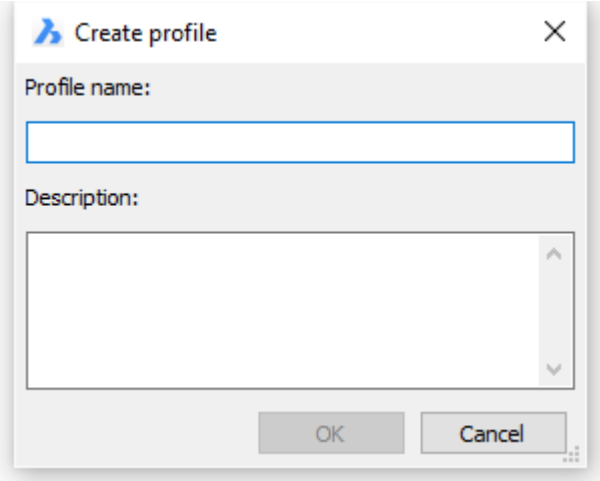
Accessing the Command

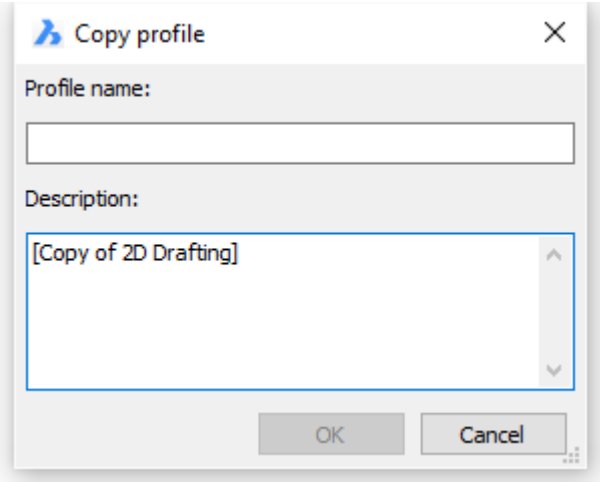
command bar: *profilemanager*
menu bar: *Tools | User Profile Manager*

: ***profilemanager***
Displays a dialog box:



Command Options

Option	Description
Create	Creates a new fresh profile, using built-in default settings. Displays a dialog box:  *Profile name*: Type a name for the new profile

	Description: (optional) Type a description for the new profile.
Set Current	Select a profile in the list, then click the *Set Current* button to set a profile current. The next time BricsCAD is started using this profile. The current profile is marked in the list.
Copy	Creates a new profile as a copy of an existing profile. Displays a dialog box:  *Profile name*: Type a name for the new profile *Description*: (optional) Type a description for the new profile.
Delete	Select a profile in the list, then click the *Delete* button to delete the profile.
Export	Select a profile in the list, then click the *Export* button to save the profile to a . Specify the export folder and filename. Exporting then importing a profile allows to migrate user preferences from one computer to another.
Import	Click the Import button, then select a **.arg* file to import a saved profile.
Start	Starts another BricsCAD session using the selected profile. Double click the profile in the list, or select the profile and click the *Start* button. If the *SingletonMode* user preference is set to *Yes* it is not possible to have multiple BricsCAD sessions open simultaneously. Use the Singleton command to set the *SingletonMode* user preference.
OK	Closes the *User Profile Manager* dialog box.

Procedures

Creating user profiles
Starting BricsCAD with a specific user profile
Restarting BricsCAD using a different user profile
Managing user profiles
To export a user profile
To import a user profile
To import a user profile from another user
Using the Welcome dialog box

Related Commands

Customize - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, aliases, and shell commands.

ProjectGeometry

Projects 2D geometry onto a region, a surface or a 3D solid; allows to create additional linework on the faces of solids and surfaces.

Accessing the Command

command bar: *projectgeometry*
menu bar: *Draw | Project Geometry*
toolbar: *Draw |*

: ***projectgeometry***
Prompts you in the command bar:

Select 2d entities to be projected or specify [PROjection direction]: (Select a 2D entity or choose an option.)

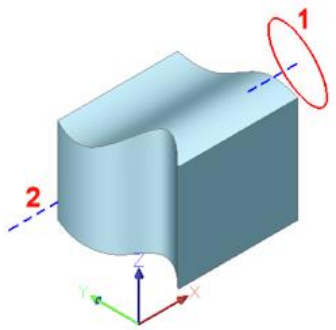
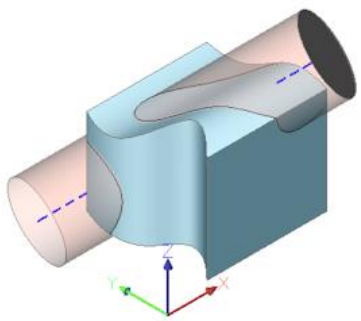
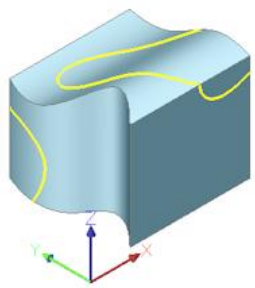
Entities in set: 1

Select 2d entities to be projected or specify [PROjection direction]: (Select more 2D entities or press Enter.)

Select a region, surface or 3d solid to project onto: (Select a region, surface or 3D solid.)

1 object(s) projected successfully.

The intersection lines of the projected entity and the 3D solid, surface or region are created on the current layer.

		
Start: 2D entity (1) projection direction (2)	*The 2D entity is projected along the projection direction.*	*Result*

Command Options

Option	Description
Projection direction	By default 2D geometry is projected orthogonally onto the Allows to define the projection direction. Prompts you: **Specify the direction [View/Ucs/Points] <View>:** • View: Sets the projection direction perpendicular to the current view orientation. • UCS: Z-axis of the current UCS. • Points: Prompts you to specify the start and end point of the projection direction.

Related Commands

Imprint - imprints 2D entities onto planar faces of 3D solids and surfaces; allows to create additional edges on planar faces.
XEdges - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Properties

Displays the Properties panel with properties of selected entities.

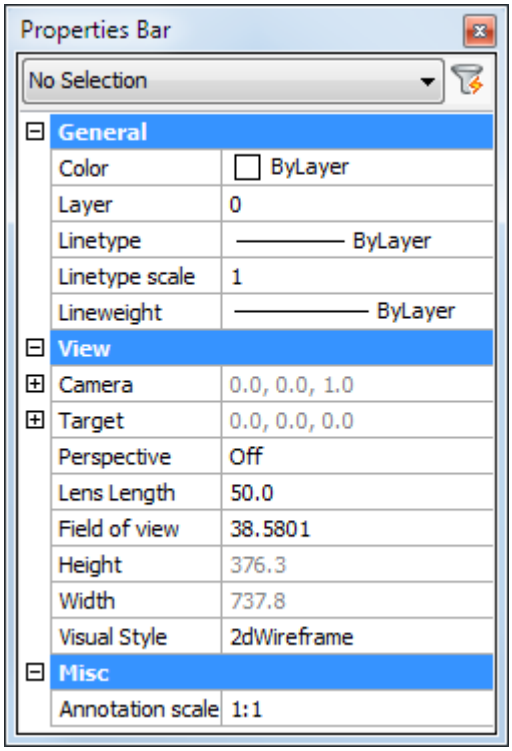
Accessing the Command

command bar: *properties*
aliases: *ch, pr, props, mo, ddchprop, ddmodify*
menu bar: *Modify | Properties*
toolbar: *Standard* |

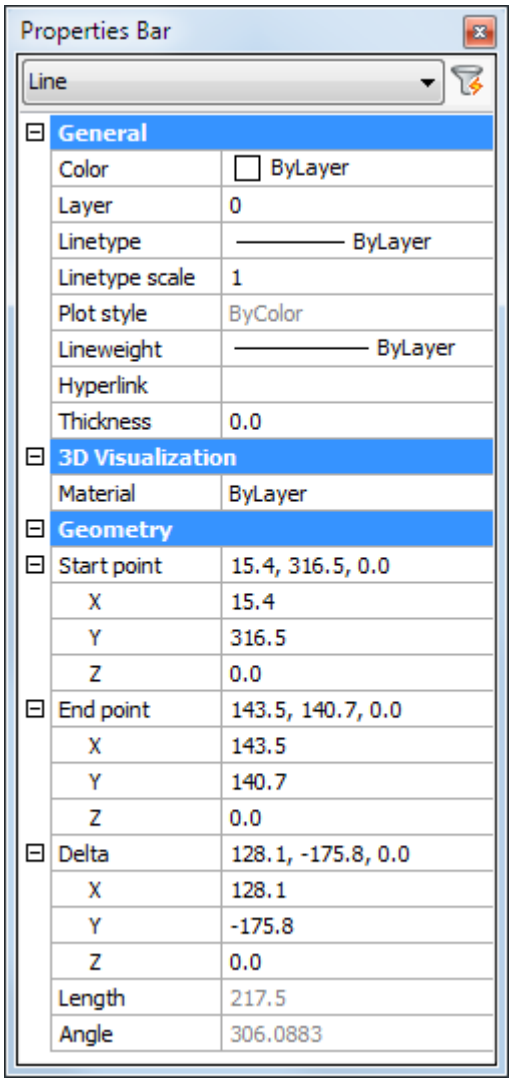
: ***properties***

Displays a palette with varying content:

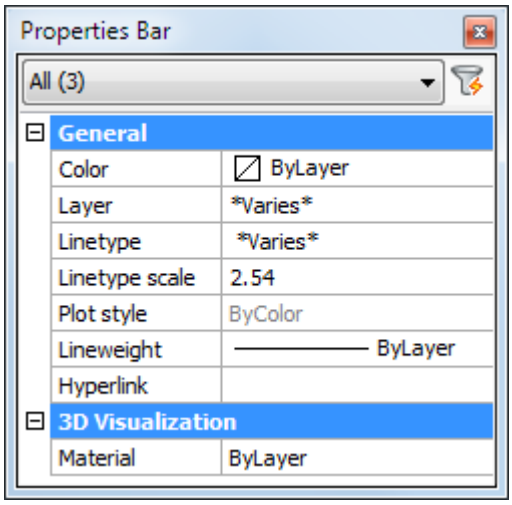
- When no entities are selected:
  General: shows the current entity creation settings.
  View: shows the current View properties (see the View command).
  Misc: shows the current Annotation scale.



- When one entity is selected: reports the properties of the selected entity.

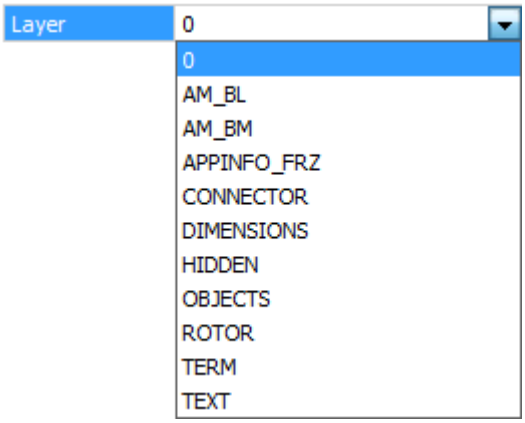
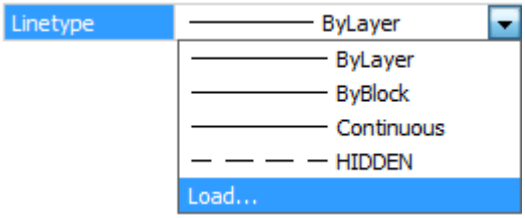
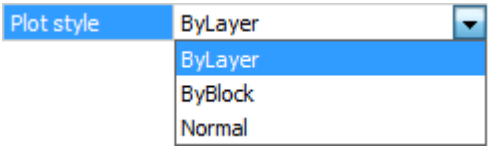


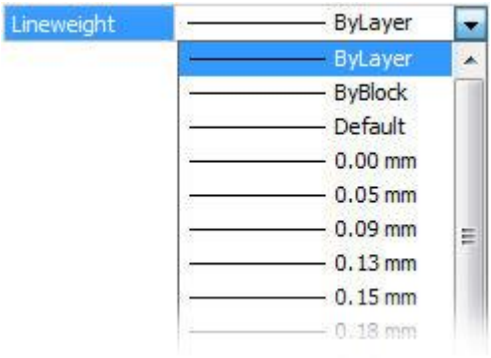
- When two or more entities are selected: reports the shared properties of the selected entities.



Command Options

Options	Description
All (1188)	Reports the number of selected entities. Click to see the number of each entity type in the current selection set: All (1188) 2D Polyline (18) Block Reference (8) Circle (10) Diametric Dimension Radial Dimension Rotated Dimension (104) Hatch (62) Leader Line (640) Polyline (320) Text (23) Select an entity type in the list to see the shared properties of all entities of this type in the current selection set.
Quick Select	Switches between *Properties* and *Quick Select* mode (See also the Qselect command).
General	
Color	Reports the current color. Allows to change the color of selected entities: 1. Click the droplist and then choose another color. 2. Choose Select Color to access the Colors dialog box. Color: ByLayer ByLayer ByBlock Red Yellow Green Cyan Blue Magenta White 249 Select color... See Color command.
Layer	Reports the current layer name. To change the layer of selected entities: 1. Click the droplist, and then choose another name.

	 See Layer command.
Linetype	Reports the current linetype. To change the linetype of selected entities: 1. Click the droplist. and then choose another name.  2. Choose Load to access the Load Linetypes dialog box. See **Linetype** command.
Linetype Scale	Reports the current linetype scale factor. To change the linetype scale, enter a new value. See LTSCALE system variable,
Plot Style	Reports the current plot style. To change the plot style of selected entities: 1. Click the droplist and then choose another name.  (This property is available only when plot styles are enabled for the drawing. See PlotStyle command.)
Lineweight	Reports the current lineweight. To change the lineweight of selected entities: 1. Click the droplist and then choose another width.

	 See LWeight command.
Hyperlink	Reports the current hyperlink. To change the hyperlink of selected entities: 1. Enter a new URL. 2. Click the button to access the Edit Hyperlinks dialog box. See Hyperlink command.
Thickness	Reports the current entity thickness. To change the thickness of selected entities: 1. Enter a new value. See **Elev** command.
Material	Reports the current Material definition of the selected entity. To change, click the settings field, then select a material from the drop-down list. See the Materials command to manage material definitions.
View	
Camera / Target	Reports the current position of the Camera and Target properties of the view. See the Camera command.
Perspective	Reports the current value of the PERSPECTIVE system variable. Allows to switch the perspective view mode on/off.
Lens Length	Reports the current value of the LENSLENGTH system variable. Allows to edit the value of the LENSLENGTH system variable.
Field of view	Reports the current view angle. Allows to edit the camera view angle. Editing this value affects the value of the LENSLENGTH system variable.

Height / Width	Reports the dimensions of the current view in drawing units.
Visual Style	Reports the current visual style. Allows to select a different visual style. See the VisualStyles command.
Misc (Miscellaneous)	
Annotation Scale	Reports the current value of the CANNOSCALE system variable. Allows to edit the value of the CANNOSCALE system variable. All scales defined in the drawing are available (see the ScaleListEdit and -ScaleListEdit commands)

Other properties vary, depending on the entity selected.

Procedures

To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

Area - reports the area, circumference, and perimeter of entities.
AttEdit - edits attribute values and properties.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
Dist - reports the 2D and 3D distances between two points.
EAttEdit - edits attribute values and most properties.
Id - reports the x,y,z coordinates of picked points.
Layer - assigns properties to layers.
List - lists the area and other parameters of entities in the command bar.
MassProp - reports the properties of solid models and regions.
PropertiesClose - closes the Properties panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

PropertiesClose

Closes the Properties panel.

Accessing the Command

command bar: *propertiesclose*
alias: *prc*

: ***propertiesclose***

Closes the Properties panel.

To open the Properties palette, use the Properties command.

Command Options

No options.

Related Command

Properties - opens the Properties panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

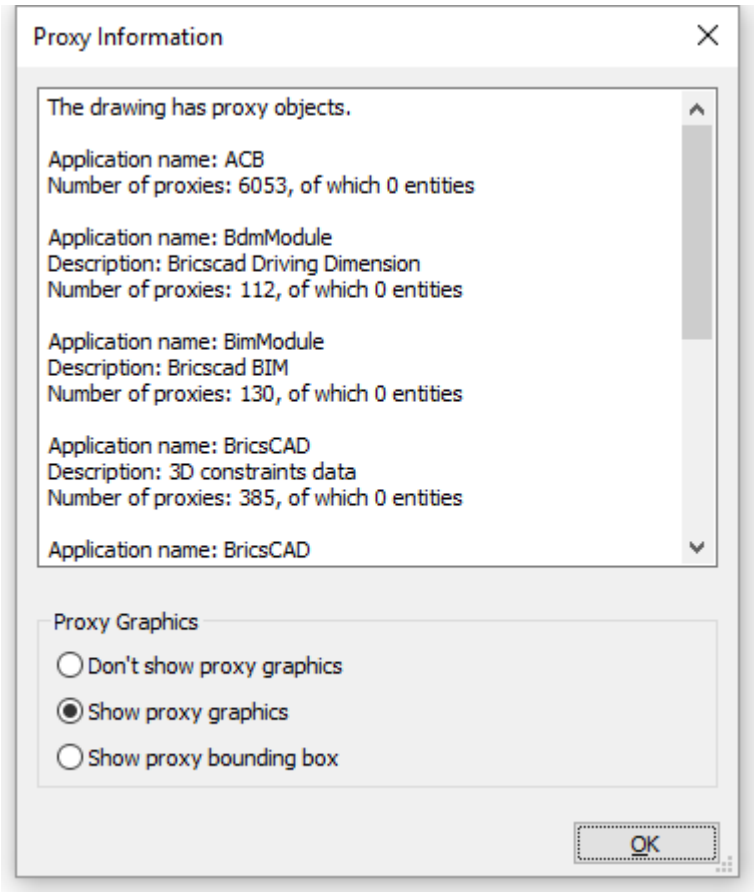
ProxyInfo

Displays a information about proxy entities, contained in the current drawing, in a dialog box.

Accessing the Command

command bar: *proxyinfo*

: ***proxyinfo***
Displays a dialog box:



Command Options

No command options.

Related Commands

Audit - repairs drawings that are open.
Recover - recovers damaged drawings.

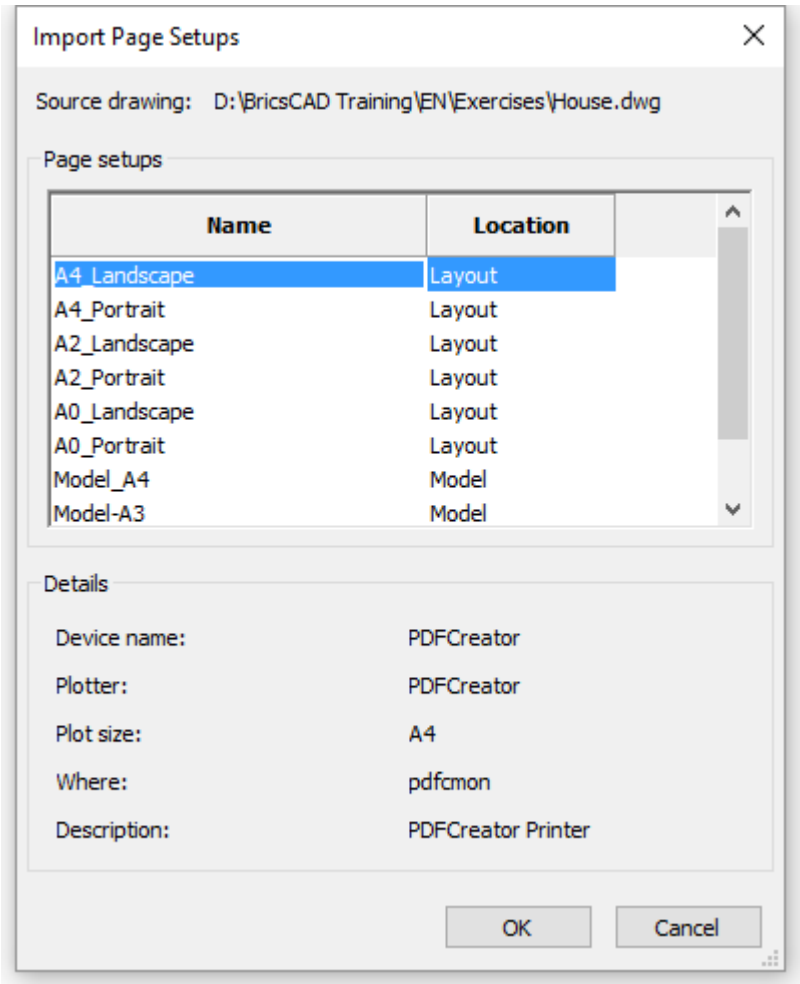
PsetupIn

Imports page setup definitions from another drawing through a dialog box.

Accessing the Command

command bar: *psetupin*

: ***psetupin***
Displays a File dialog.
(Select the drawing you want to import page setups from.)
Displays a dialog:



(Select one or more page setups in the list. Press and hold the Ctrl key to select multiple page setups, then press the OK key to import the selection into the current drawing.)

Command Options

No options.

Related Commands

-PsetupIn - imports page setup definitions from another drawing through the command bar.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.

Layout - creates, copies, renames, and deletes layouts.

-PsetupIn

Imports page setup definitions from another drawing through the command bar.

Accessing the Command

command bar: *-psetupin*

: ***-psetupin***

Displays a File dialog.

(Select the drawing you want to import page setups from.)

Prompts you in the command bar:

Enter user defined page setup(s) to import or [?]: (Type the name(s) of the page setup(s) you want to import - separate setup names by commas - or type ? to list the available page setups.)

Command Options

No options

Related Commands

PsetupIn - Imports page setup definitions from another drawing through a dialog box.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.

PSpace

Switches from model to paper space in a layout tab (short for "paper space").

Accessing the Command

command bar: *pspace*
shortcut: *Double-click outside a layout viewport.*
alias: *ps*
status bar: *M:Layout*

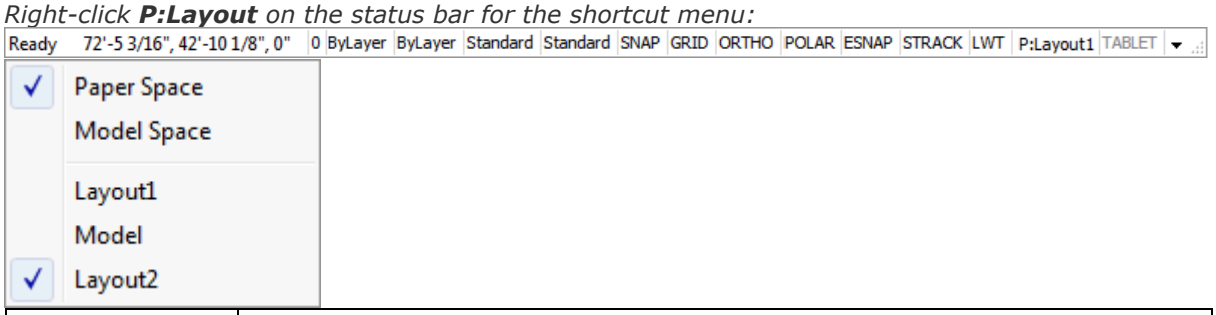
: ***pspace***
Displays no prompts in the command bar.

Command Options

No options.

Status Bar Menu

*Right-click **P:Layout** on the status bar for the shortcut menu:*



Shortcut Menu	Description
Paper Space	Switches to paper space in the current layout.
Model Space	Switches to model space in the current layout.
Layout1	Switches the drawing editor to Layout1.
Model	Switches the drawing editor to Model.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Related Commands

ChSpace - Moves entities from paper space to model space and vice versa.
MSpace - switches to model space.
MView - creates viewports in paper space.
VpClip - clips viewports.
VPorts - creates viewports in model space.
Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

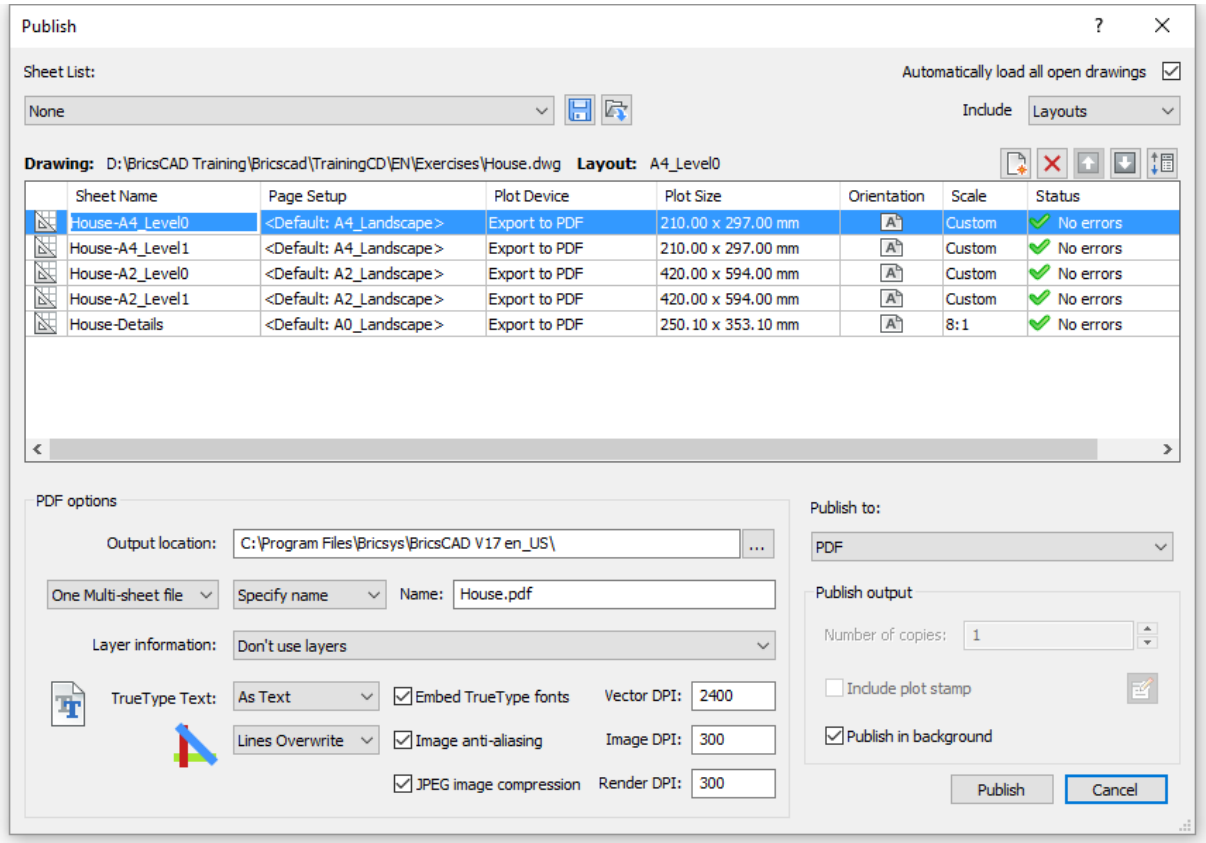
Publish

Allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

Accessing the Command

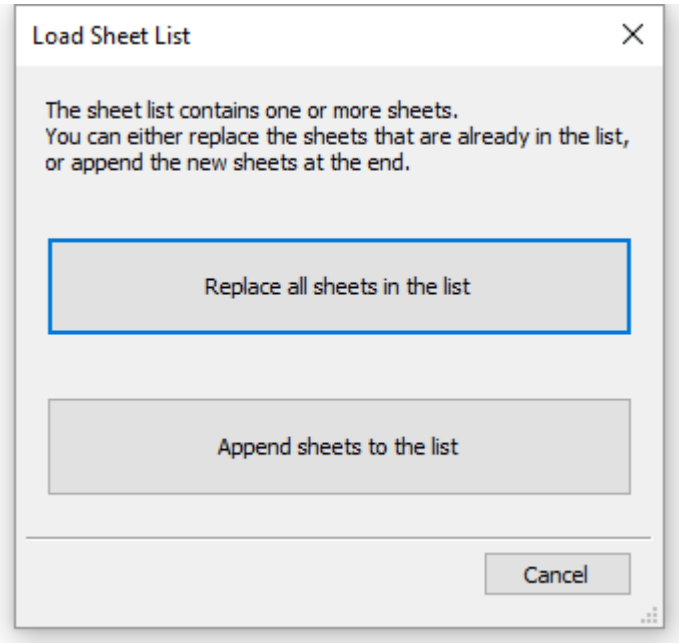
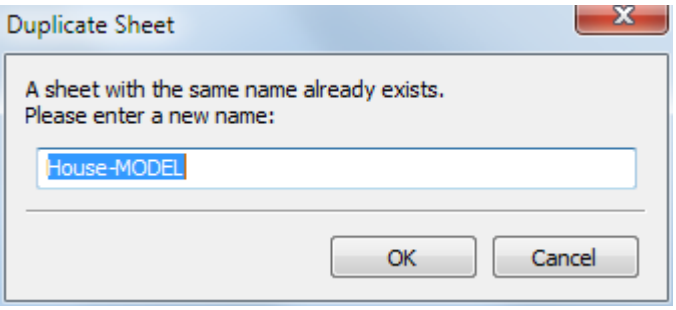
command bar: *publish*
menu bar: *File | Publish*
toolbar: Standard |

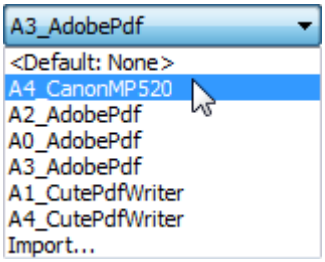
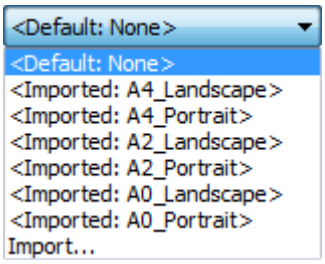
: ***publish***
Displays a dialog box:

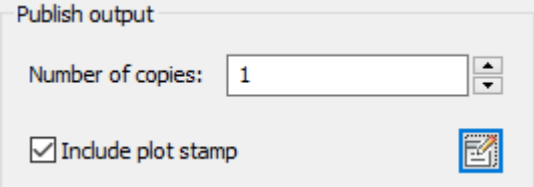
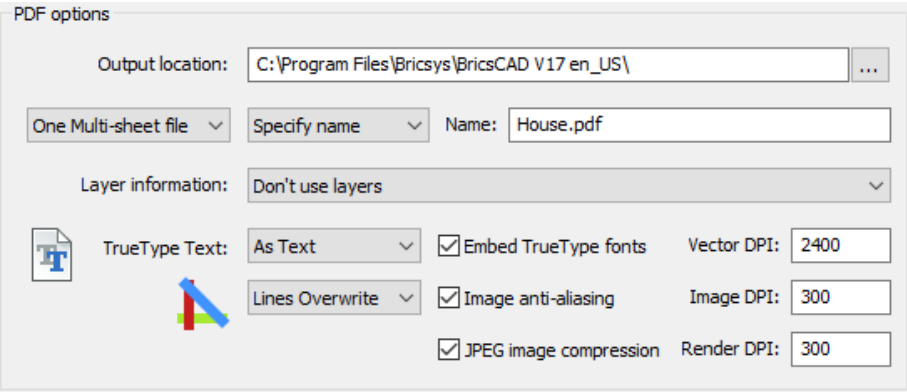


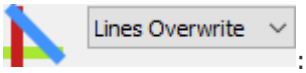
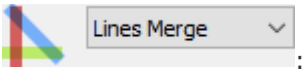
Command Options

Option	Description
Load Sheet List	Loads a saved sheet list. If the current list is not saved yet, you are prompted to save it. A File dialog displays. Select a *.dsd (Drawing Set Description) file, then double click or click the *Open* button. Choose whether to *Replace* the current sheet list or to *Append* the sheets in the *.dsd file to the current sheet list.

	
Save Sheet List	Saves the current sheet list to a a *.dsd (Drawing Set Description) file.
Automatically load all open drawings.	If checked, adds the model and/or paper space layouts of all open drawings. If not checkked, adds the model and/or paper space layouts of the current drawing only. Sets the value of the PUBLISHALLSHEETS system variable.
Include	Determines which sheets must be included. The options are: *Model*, *Layouts*, *Model & Layouts*
Add Sheets	Adds sheets to the current list. Displays a *File* dialog. Select a drawing. All layouts in the selected drawing are added to the sheet list. If a sheet name already exists, a dialog box displays prompting you to enter a new name. 
Remove Sheets	Deletes the selected sheet(s) from the current list.

Arrange Sheets	Allows to move selected sheets up or down or to reverse the sheet order.
Drawing	Path and name of the drawing of the currently selected sheet.
Layout	Layout name of the currently selected sheet
Sheet List	
Sheet Name	Layout name of the sheet.
Page Setup	Page Setup name of the sheet. Click the down arrow on the list button to select a different page setup.  Choose *Import...* to import page setups of another drawing. A File dialog displays: select a drawing, then double click or click the *Open* button. All page setups in the selected drawing are available in the *Page Setup* list button. 
Plot Device	Name of the plot device as defined in the Page Setup of the sheet.
Plot Size	Size of the paper in the currently selected page setup.
Orientation	Paper orientation in the currently selected page setup: Landscape ( ) or Portrait ( ).
Scale	Scale of the currently selected page setup.
Status	Status of the currently selected page setup.

Publish to:	Allows to choose the output target of the Publish command:
	• Plotter named in page setup.
	 
	
	  • Number of copies: sets the number of copies for each sheet to be printed.
	  • Include plot stamp: if checked adds a plot stamp. Click the *Edit Plot Stamp* button ( ) to change plot stamp data.
	• PDF
	TUTORIALS
	Output as PDF
	 
	
	  • Output location: Click the Browse button ( ) and select a folder.
	  • File: Select an option:
	  • Single-Sheet files: each layout is published to a separate pdf. The file names are: <drawing>_<layout>.pdf
	  • One Multi-sheet file: layouts are published to a single file. You can choose to specify a name or to be prompted for a name when the publish procedure starts.
	  • 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:
	  • TrueType text can be either exported as text or converted to geometry. The latter option ensures that the text in the PDF looks exactly as in the drawing. Converting to geometry

	increases the file size and pixelation might occur when viewed at a high zoom level. • Embed TrueType fonts: If the font is not embedded, a substitute font might be used by the PDF viewer. Please notice that licensed fonts cannot be embedded. • Merge Control: Determines whether overlapping lines are merged or lie on top of each other. •  : the lines on top overwrite the lines below. •  : the colors of crossing lines are blended into a new color. • Image anti-aliasing: If the resolution of an image is too low to match the *Image DPI* setting then the raster image is anti-aliased (generate pixels artificially) to make the raster image look more smoothly. Without that setting an image with a too low resolution is embedded "as-is". • 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 to lower the resolution of the original raster image in order 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. • Render DPI: Defines the resolution of the resulting image in the PDF for rendered viewports (3D visual styles). This setting is used to generate the rendered view image at that resolution specific for the PDF.
Publish in background	The BACKGROUNDPLOT system variable controls whether documents are published in the current session or in a BricsCAD session running in the background, which allows to continue working while printing.
Publish	Publishes the current sheet set to the selected output target.

Related Commands

ExportPdf - exports layouts in a drawing to a pdf docucment.

-Publish - prints a saved sheet list (= a list of model space or paper space layouts).

PageSetup - assigns plot style to drawings.
Plot - plots drawings through the command bar.
Print - plots drawings through a dialog box.
SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.

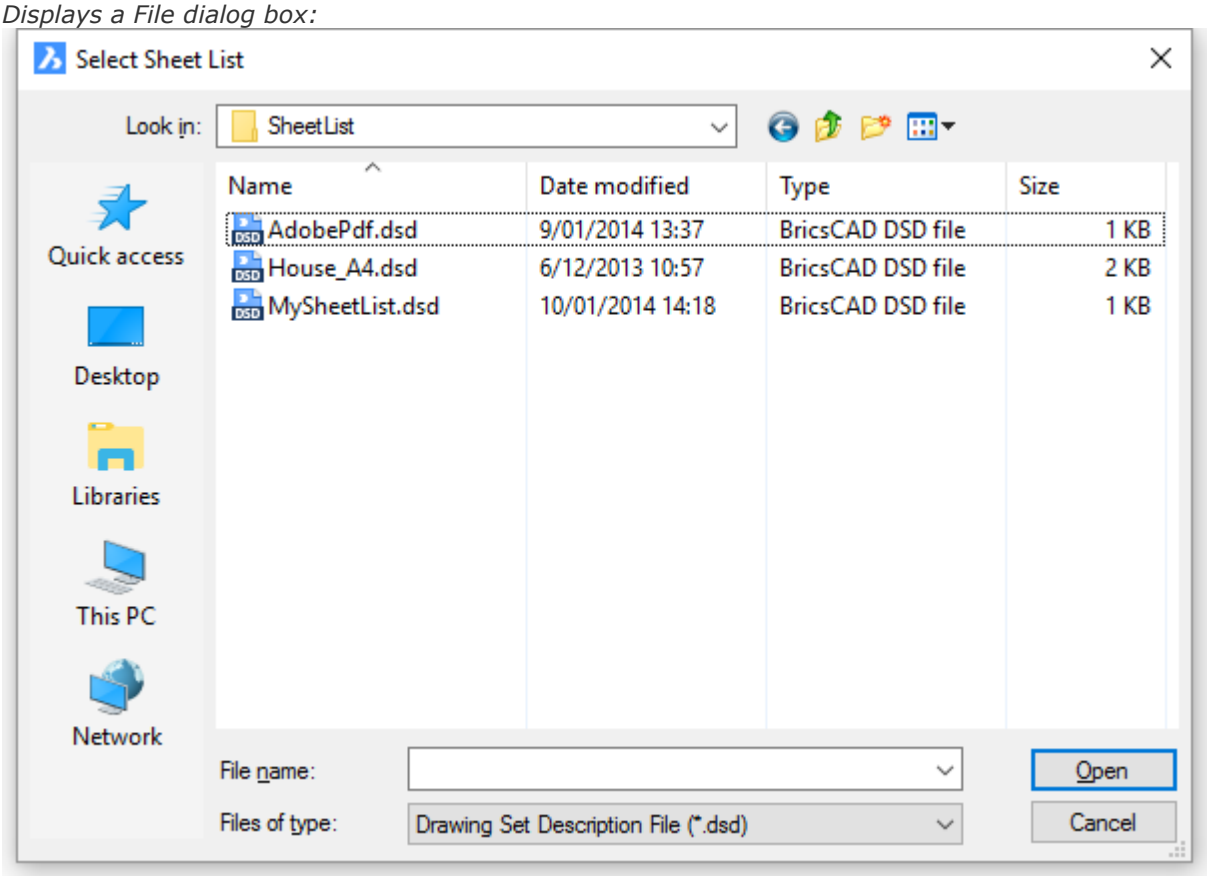
-Publish

Prints a saved sheet list (= a list of model space or paper space layouts).

Accessing the Command

command bar: *-publish*

: ***-publish***
Displays a File dialog box:



Select a file in the list, then click the Open button.

If the BACKGROUNDPLOT system variable is 2 or 3, the sheets are printed in the background.

Command Options

No options.

Related Commands

ExportPdf - Exports layouts in a drawing to a pdf docucment.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.
PageSetup - assigns plot style to drawings.
Plot - plots drawings through the command bar.
Print - plots drawings through a dialog box.
SheetSet - manages sheet sets. Opens the *Sheet Sets* panel.

Purge

Removes unused named entities from drawings, such as unused layers and linetypes.

Accessing the Command

command bar: *purge*
alias: *pu, -purge*
menu bar: *File | Purge*

: ***purge***
Prompts you in the command bar:
Purge: [BAtch all/purge All/Blocks/Dimension styles/LAyers/LineTypes/MAterials/MLinestyles/Plotstyles/Regapps/text STyles/Tablestyles/Visualstyles/Empty text entities/Orphaned data]: *(Enter an option.)*

Command Options

Option	Description
baTch all	Purges all unused named entities from the drawing without prompts.
purge All	Purges all unused named entities from the drawing with the following prompt: Purge *name*? (Yes/No/yes to All)? <No> - enter an option. • Yes - purges the named entity. • No - does not purge the named entity. • All - purges all remaining unused named entities.
Blocks	Removes unused block definitions from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of block definitions to purge, or press Enter to purge all unused ones.
Dimension styles	Removes unused dimension styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of dimension styles to purge, or press Enter to purge all unused ones. Dimstyle Standard cannot be purged.
LAyers	Removes unused layers from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of layers to purge, or press Enter to purge all unused ones. Layer 0 cannot be purged.
LineTypes	Removes unused linetypes from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of linetypes to purge, or press Enter to purge all unused ones. Linetype Continuous cannot be purged.
MAterials	Removes unused materials from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of materials to purge, or press Enter to purge all unused ones. The Global material cannot be purged.
MLinestyles	Removes unused multiline styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of multiline styles to purge, or press Enter to purge all unused ones. Style Standard cannot be purged.

Plotstyles	Removes unused plot styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of plot styles to purge, or press Enter to purge all unused ones.
Regapps	Removes unused regapp keys (1) from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names regapp keys to purge, or press Enter to purge all unused ones.
text STyles	Removes unused text styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of styles to purge, or press Enter to purge all unused ones. Style Standard cannot be purged.
Tablestyles	Removes unused table styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of table styles to purge, or press Enter to purge all unused ones. Style Standard cannot be purged.
Visualstyles	Removes unused visual styles from the current drawing; prompts you: Enter name(s) to purge <*> - enter the names of visual styles to purge, or press Enter to purge all unused ones. User defined visual styles can be purged only. Hard-coded visual styles cannot be purged.
Empty text entities	Removes empty text entities from the current drawing.
Orphaned data	Performs a drawing scan and removes obsolete DGN linestyle data.

(1) Regapp keys are used to attach additional information to entities in a drawing, e.g. by 3rd party applications, which is saved with the drawing. Regapp is short for 'Registered applications'.

Related Command

Explorer - purges unused entity names through the Drawing Explorer.

Pyramid

Creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)

Accessing the Command

command bar: *pyramid*
alias: *pyr*
menu bar: *Model | 3D Solids | Pyramid*
toolbar: *3D Solids |*

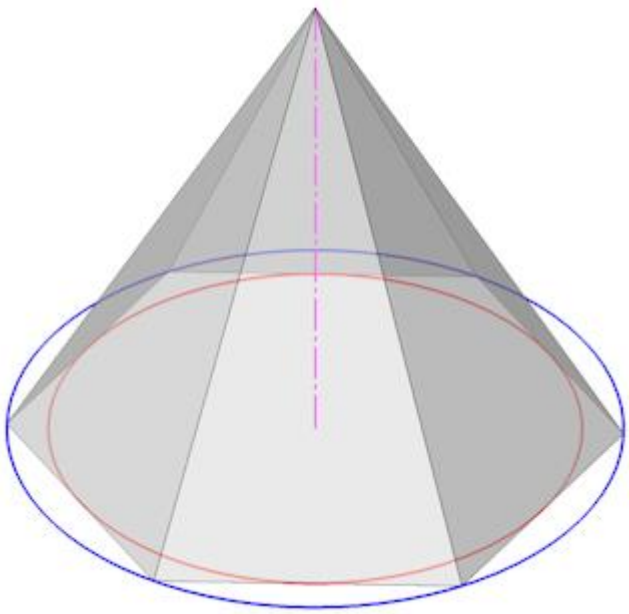
: ***pyramid***

Command sequence in Pro and Platinum

Prompts you in the command bar:
Specify center point or [Edge/Sides]: (Pick a point, enter coordinates or choose an option.)

Specify base radius or [Inscribed]: (Type a value and press Enter, pick a point or choose an option.)

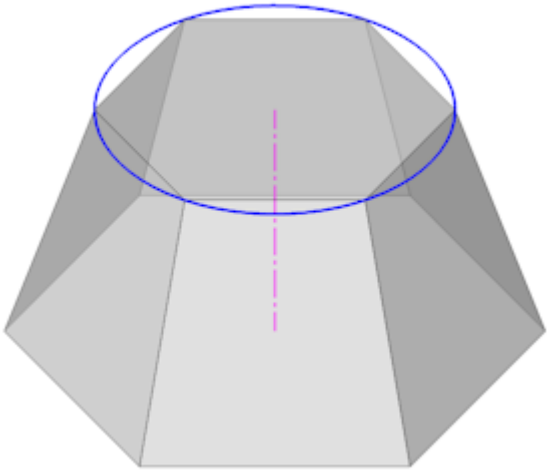
Specify height or [2Point/Axis endpoint/Top radius]: (Type a value and press Enter, pick a point or choose an option.)



The base face of the pyramid is inscribed in the blue circle and circumscribes the red circle.

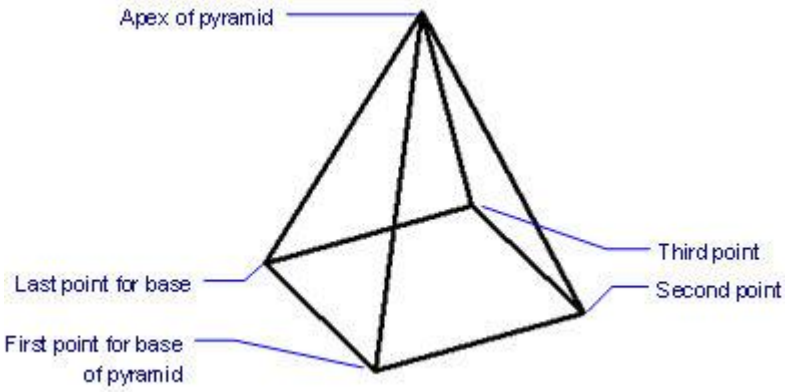
Command Options

Option	Description
Sides	Allows to specify the number of sides or vertices of the base face. Prompts you: Enter number of sides <4>: Type a number of press Enter to accept the current number.
Edge	Allows to specify the length of the edges of the base face. Prompts you: Specify first endpoint of edge: specify a point. Specify second point of edge: specify a point.

Base radius Inscribed	Allows to define the radius of the circle the base face of the pyramid is inscribed in. Prompts you: Specify base radius or Circumscribed: enter a value or specify a point.
Base radius circumscribed	Allows to define the radius of the circle the base face of the pyramid circumscribes. Prompts you: Specify base radius or Inscribed: enter a value or specify a point.
2Point	Allows to set the height of the pyramid by specifying two points. Prompts you: Specify first point: **specify a point.** Specify second point: specify a point
Axis endpoint	Allows to set the length and orientation of the pyramid axis by specifying the top of the pyramid, creating a rotated pyramid. Prompts you: Specify axis endpoint: **specify a point.**
Top radius	Allows to create a truncated pyramid. Prompts you: Specify top radius <0.0>: type a value or specify a point to define the radius of the circle the top face of the pyramid is inscribed in. Specify height or **2Point**/**Axis endpoint**: type a value or specify a point. 

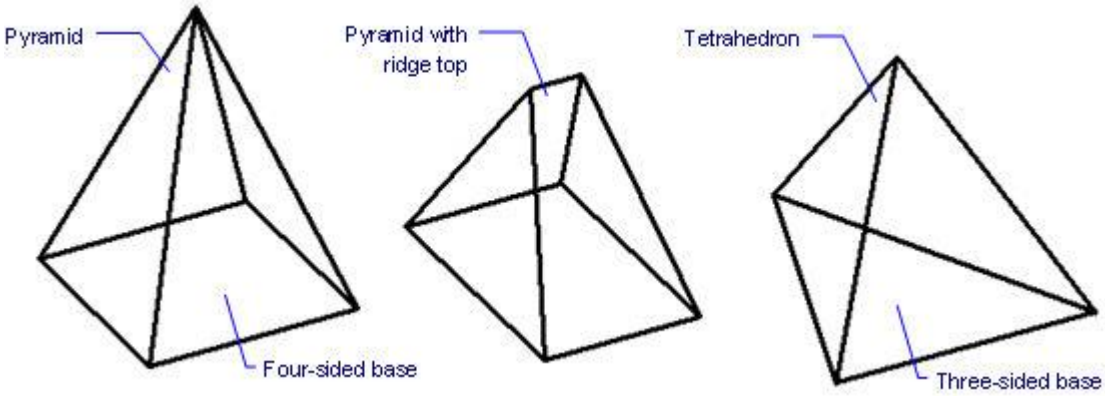
Command sequence in Classic

Prompts you in the command bar:
First point for base of pyramid: (Pick a point, or enter coordinates.)
Second point: (Pick another point, or enter coordinates.)
Third point: (Pick a third point, or enter coordinates.)
Tetrahedron/<Last point for base>: (Pick the last point, enter coordinates, or type T.)
Set apex of pyramid or [Ridge/Top surface]:
Ridge/Top surface/<Apex of pyramid>: (Enter an option, pick a point, or enter coordinates.)



This command draws:

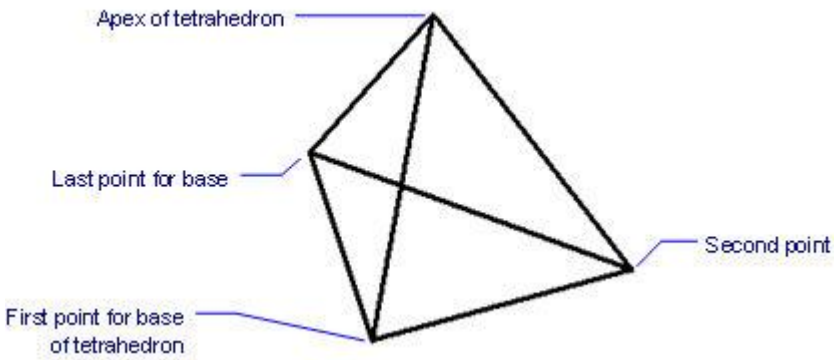
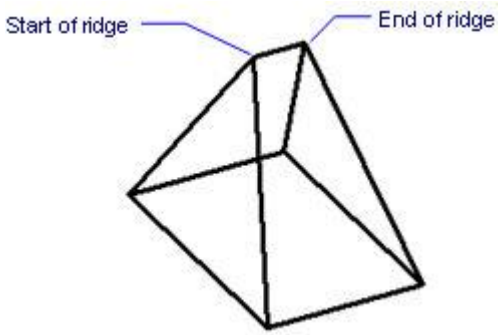
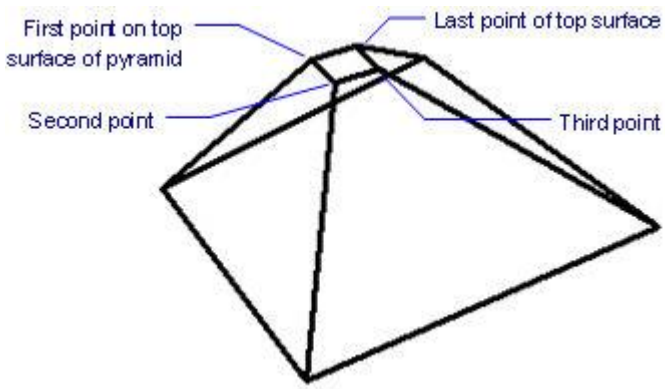
- Pyramids with four-sided base and four sides; the base does not need to be an equilateral.
- Tetrahedrons with triangular base and three sides; the base can be scalene.
- Pyramids with pointed, ridged, or flat tops.



To draw a rotated pyramid, specify the angle during the Apex option.

Command Options

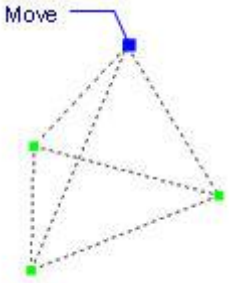
Option	Description
First point for base of pyramid	Specifies the first corner of the pyramid's base.
Second point	Specifies the second corner of the base.
Third point	Specifies the third point of a pyramid's base, or the last point for a tetrahedron base.
Last point for base	Specifies the fourth point for a pyramid's base.
Apex of pyramid	Specifies the height and angle of the pyramid; draws a pointy top.

Tetrahedron  	Draws tetrahedrons. Prompts you: Apex of tetrahedron - specifies the top of the tetrahedron.
Ridge  	Draws a ridge at the top of the pyramid. Prompts you: Start of ridge - specifies the height and location of one end of the ridge. End of ridge - specifies the height and location of the other end of the ridge.
Top surface  	Draws a flat surface at the top of the pyramid. Prompts you: First point on top surface of pyramid - specifies the starting corner of the pyramid's flat top. Second point - specifies the next corner. Third point -specifies the third corner for a pyramid, and the last corner for a tetrahedron. Last point of top surface - specifies the fourth corner for the pyramid.

Grips Editing

Pyramids can be edited directly through grips:

1. Select the pyramid. Notice that it has several grips.
2. Drag a grip to move the pyramid. (All grips perform the same function: move.)



Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.

Q

QLeader

Draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

Accessing the Command

command bar: *qleader*
menu bar: *Dimension | QLeader*
toolbar: *Dimensions |*

: ***qleader***
Prompts you in the command bar:

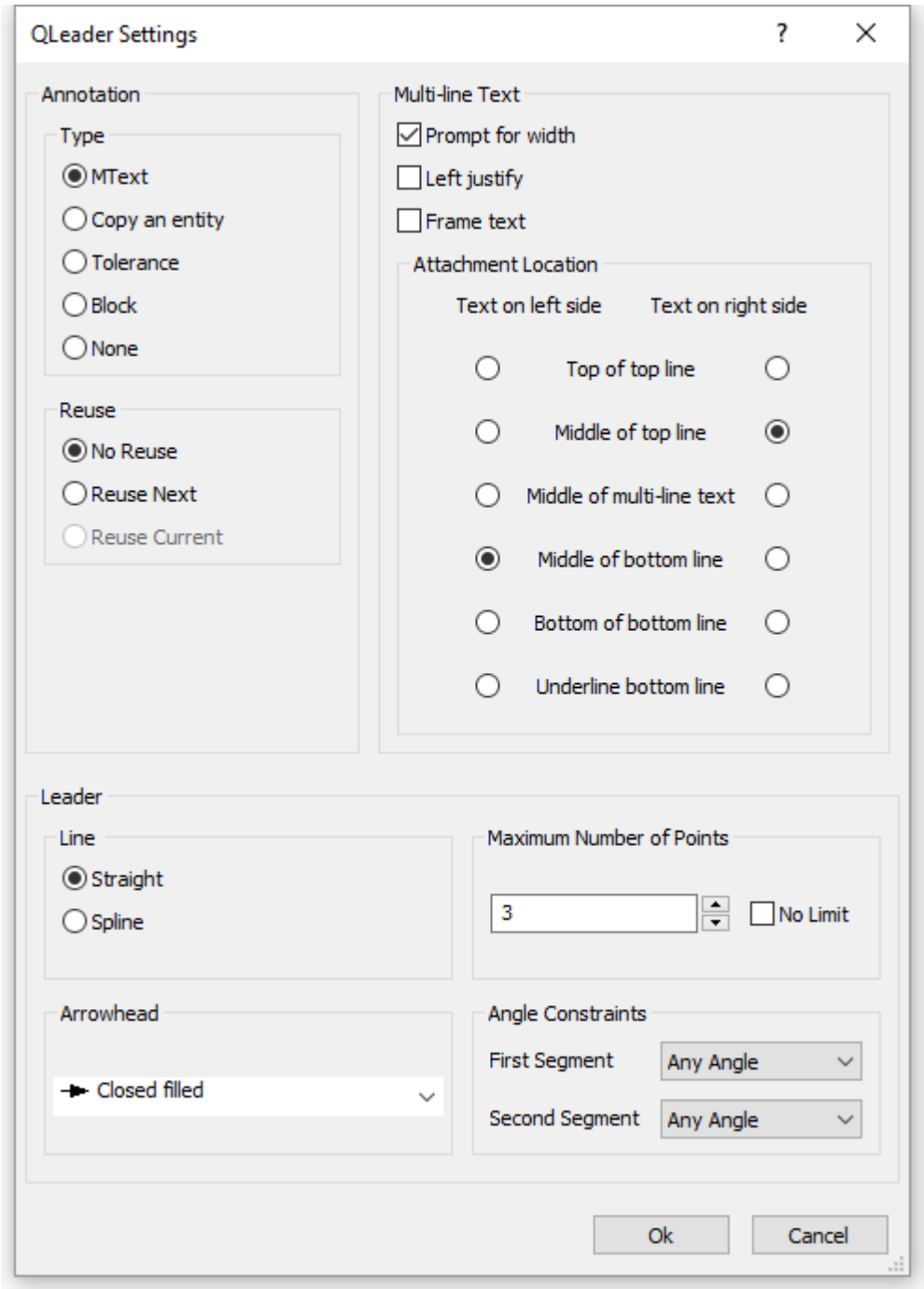
Start of leader <Settings>: (Pick a point press Enter to display the *Qleader Settings* dialog box.)

Next point: (Pick another point)

To point: <Undo>: (Pick another point or press Enter to undo the last leader segment.)

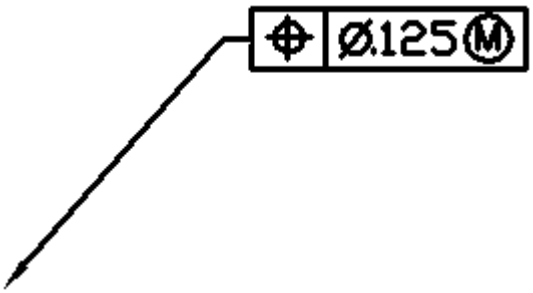
First line of annotation text: (Type the first line of the annotation text.)

Next line of annotation text: (Type the second line of the annotation text or press Enter to exit the command.)



Command Options

Option	Description
Annotation type	Specifies the type of annotation.
Mtext	Starts adding text to the end of the leader line. The leader can have one or more lines of text.

Copy of an entity	Uses any other entity in the drawing as the annotation; prompts you: **Select an entity** - choose an entity, other than the leader itself.
Tolerance	Uses tolerance notation for the annotation; displays the Geometric Tolerance dialog box.  See Tolerance command.
Block	Uses a block as the annotation; displays prompts identical to that of the -Insert command: **? to list blocks in drawing/~ to open the file dialog/<Block to insert>** - enter an option. See -Insert command for details on these options.
None	The annotation is kept blank.
Multi-line Text	Specifies some multi-line text options.
Attachment location	Specifies the placement of the annotation text.
Reuse	Specifies how to reuse the annotation.
No reuse	Do not reuse the previous annotation text. You are prompted to enter new text.
Reuse next	You are prompted to enter new text, which is then reused in the next leader(s).
Reuse current	The current text is reused for each new leader. You are not prompted to enter new text.
Leader line	Specifies the leader line: straight line segments or spline
Maximum number of points	Allows to specify the number of points for the leader.
Arrowhead	Choose an arrowhead from the drop-down list.
Angle Constraints	Allows to set an angle constraint for the first and/or the second leader segment. Click the option button, then choose an option: *Any Angle*, *Horizontal*, *90°*, *45°*, *30°*, *15°*
OK	Closes the *Qleader Settings* dialog box; prompts you to specify the first point of the leader.

Related Commands

Leader - places leaders with text, blocks, entities, or tolerances.
DimLeader - places leaders with text, blocks, entities, or tolerances.
MLeader - creates multileader entities using the current multileader style.

QNew

Opens new drawings in BricsCAD (short for "quick new").

Accessing the Command

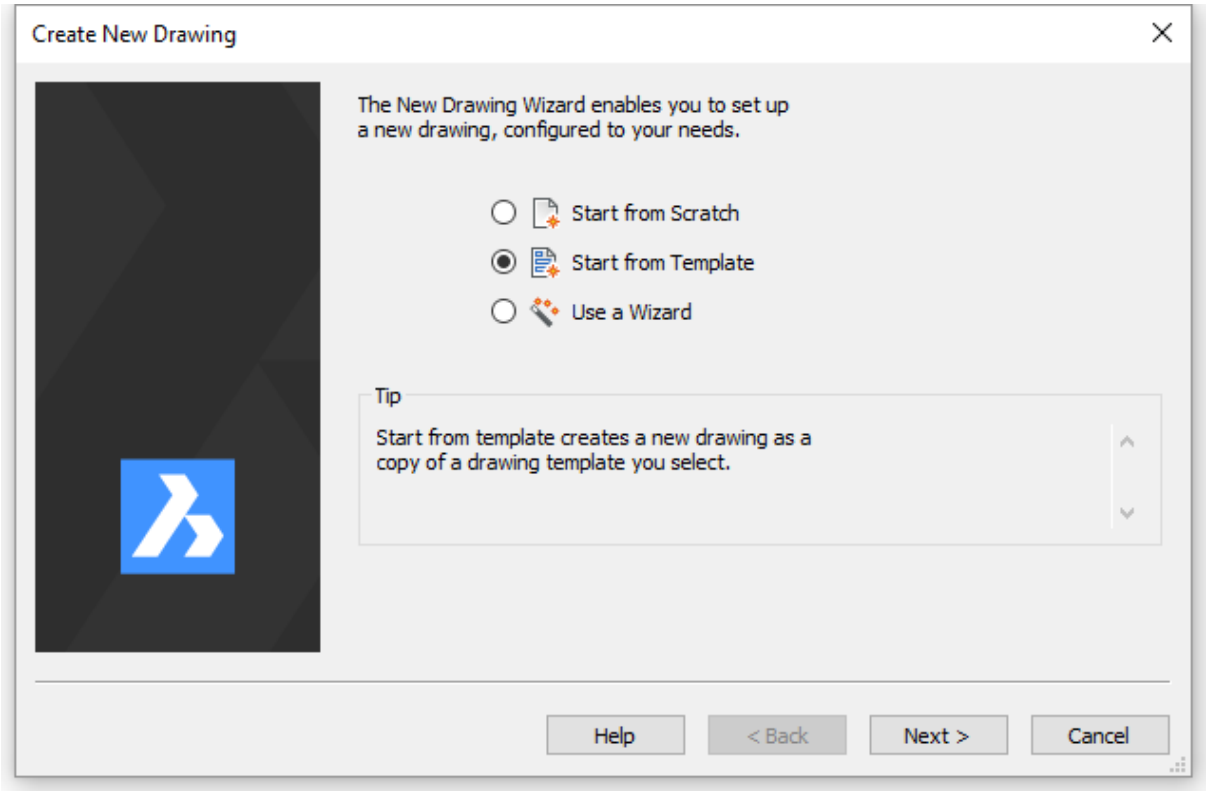
command bar: *qnew*
alias: *n*
menu bar: *Name*
toolbar: *Standard* |

: ***qnew***

Displays a dialog box that depends on the STARTUP system variable in the Settings dialog box:

When STARTUP = 1 (Display the Startup and Create new Drawings dialog boxes), shows the following dialog box if the BASEFILE* system variable is undefined.

* The BASEFILE system variable defines the default template drawing.



See NewWiz command.

When STARTUP = 2 (Display the Select Template dialog box, or use default drawing template file):

- Creates a new drawing using the default template drawing defined by the BASEFILE system variable.
- If the BASEFILE system variable is empty, displays the Select Template dialog box showing the content of the folder defined by the TemplatePatch user preference.

See New command.

Command Options

For Create New Drawing options, see NewWiz command.

For Select Template options, see New command.

Procedures

To create a drawing using a default template drawing
To set the default template drawing
To create a drawing from scratch
To set the Plot style policy
To create a drawing using a template drawing
To create a drawing using the New command
To set the Template Path variable
To create a drawing using a wizard

Related Commands

Units - sets units through the Settings dialog box.
-Units - sets units through the command bar.
NewWiz - sets units and other parameters for new drawings.
New - opens new drawings.

QPrint

Prints the drawing with the default plot configuration, without displaying the Print dialog box (short for "quick print").

Accessing the Command

command bar: *qprint*
toolbar: *Standard* |

: ***qprint***
Displays no prompts in the command bar.
To configure the plot/print, use the PageSetup command.

Command Options

No options.

Related Commands

Print - plots drawings through a dialog box.
Plot - plots drawings through the command line.
Preview - previews drawings before they are plotted.

PageSetup - assigns plot style to drawings.

PlotStyle - sets the current plot style.
StylesManager - creates and attaches plot styles to drawings.

QSave

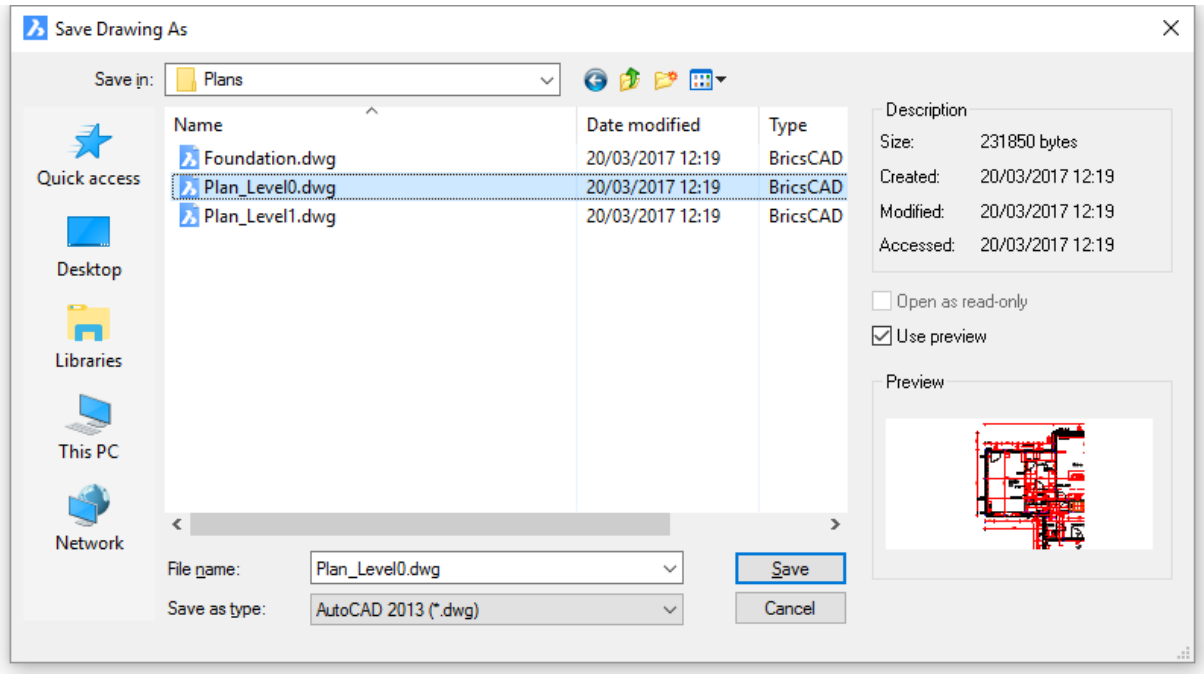
Saves the drawing without displaying the Save dialog box (short for "quick save").

Accessing the Command

command bar: *qsave*
menu bar: *File | Save*
toolbar: *Standard |*

: ***qsave***

If the drawing has never been saved before, displays a dialog box:



See **Save** command.

When the drawing has been saved, displays no dialog box or prompt in the command bar.
To save the drawing by another name, use the Save command.

Command Options

No options.

Related Commands

Save - saves the current drawing through a dialog box.
SaveAll - saves all open drawings.
SaveAs - saves drawings by another name.
SaveAsR12 - saves drawing in older formats.
Export - exports drawings in a variety of vector and raster formats.

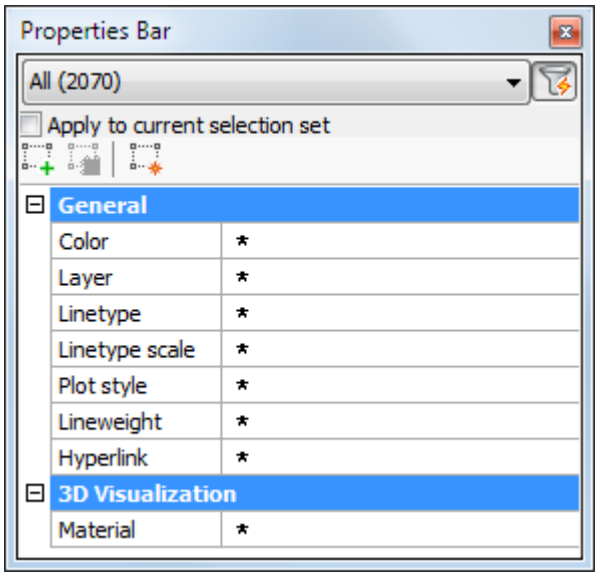
QSelect

Composes a selection set using filters.

Accessing the Command

command bar: *qselect*
toolbar: *Properties Bar* |

: ***qselect***
Switches (or opens) the Properties Bar to Quick Select mode.



Command Options

Option	Description
Apply to current selection set	Applies the filter to the current selection set.
	Adds the entities, that match the filter, to the current selection set.
	Removes the entities, that match the filter, from the current selection set.
	Adds the entities, that match the filter, to a new selection set.
	Switches the *Properties Bar* to *Quick Select* mode and back (see also the Properties command).
Filters	**Description**
All (1115)	Lists the entity types in the drawing or in the current selection set. Click the button, then choose an entity type in the list.
Entity Properties	The common properties of the entities in the drawing or in the current selection set are available. Click the property field, then do one of the following: • Choose a value from the list.

	• Type a value in the entry field. Choose an operator: * Select All = Equals ≠ Not Equal < Smaller ≤ Smaller Or Equal > Greater ≥ Greater or Equal

Procedures

Creating a selection set.

Related Commands

dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
Select - selects one or more entities; can be accessed later with the Previous option.
SelectSimilar - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE system variable
DdSelect - sets selection properties in the *Entity Selection* section of the *Settings* dialog box.
SelGrips - selects entities and displays grips.

QText

Toggles the display of text as rectangles (short for "quick text").

Accessing the Command

command bar: *qtext*
alias: *qt*

: ***qtext***
Prompts you in the command bar:

QTEXTMODE is currently off: ON/Toggle/<Off>: (Enter an option.)

The change in text does not occur until you enter the Regen command:

: *regen*

This command is generally unnecessary.

It was useful in old days when computers were slow at processing text.

Command Options

Option	Description
ON	Turns on qtext mode.
Toggle	Changes qtext mode: • If on, turns it off. • Iff off, turns it on.
Off	Turns off qtext mode.

Related Commands

Text - places single-lines of text in drawings.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
TxtExp - explodes text into polylines.
Properties - modifies the properties of text.
Find - finds and replaces text.

Quadrant

Snaps to the quadrant points of circles, arcs, and polyarcs.

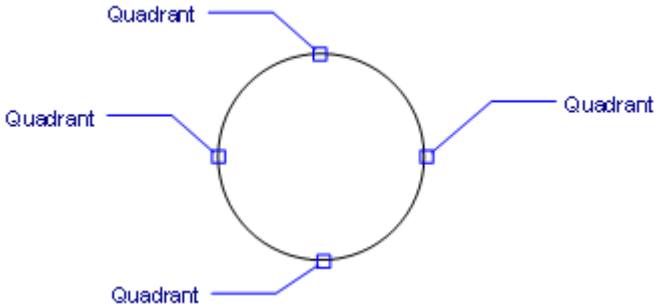
Accessing the Command

command bar: *quadrant*
transparent: *qua*
toolbar: *Entity Snaps* |

: ***quadrant***

Displays no prompts in the command bar.

The quadrant points are located at the 90-degree points of circular entities:



When quadrant snap is on, the cursor snaps to the 90-degree points of circles, arcs, and polyarcs.

Command Options

Option	Description
Quadrant	Entering this command toggles Quadrant entity snap. It snaps to the 90-degree quadrant points on curved entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.

Tangent - toggles tangent snap mode.

Quit

Ends BricsCAD; optionally saves unsaved drawings.

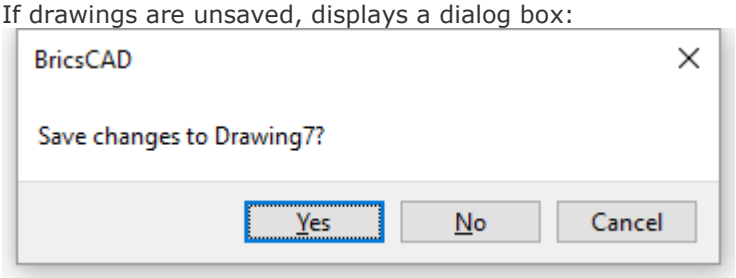
Accessing the Command

command bar: *quit*
shortcuts: *Ctrl+Alt+Q, Alt+F4*
alias: *exit*
menu bar: *File | Exit*

: ***quit***

If no drawings need to be saved, exits without displaying any dialog boxes or prompts in the command bar.

If drawings are unsaved, displays a dialog box:



Command Options

Option	Description
Yes	Saves the drawing, and then exits BricsCAD.
No	Does not save the drawing, and then exits BricsCAD.
Cancel	Cancels the command; BricsCAD does not exit.

Related Commands

Open - opens drawing files.
Close - closes the current drawing.
WClose - closes the current window.

R

Ray

Draws rays, semi-infinite lines.

Accessing the Command

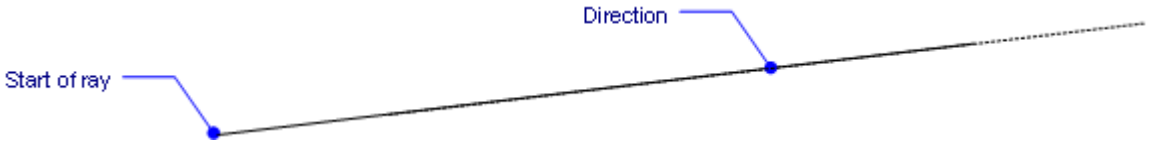
command bar: *ray*
menu bar: *Draw | Ray*
toolbar: *Draw | Line |*

: ***ray***
Prompts you in the command bar:

Infinite ray: [Bisect/Horizontal/Vertical/Angle/Parallel] <Start of ray>: *(Pick a point or enter an option.)*

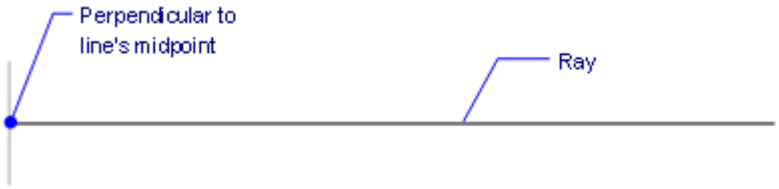
Direction: (Pick a point, or enter x,y coordinates.)

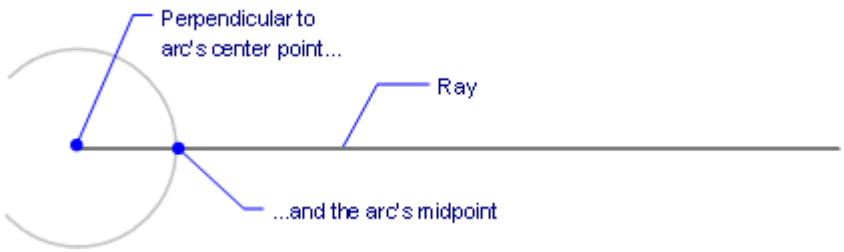
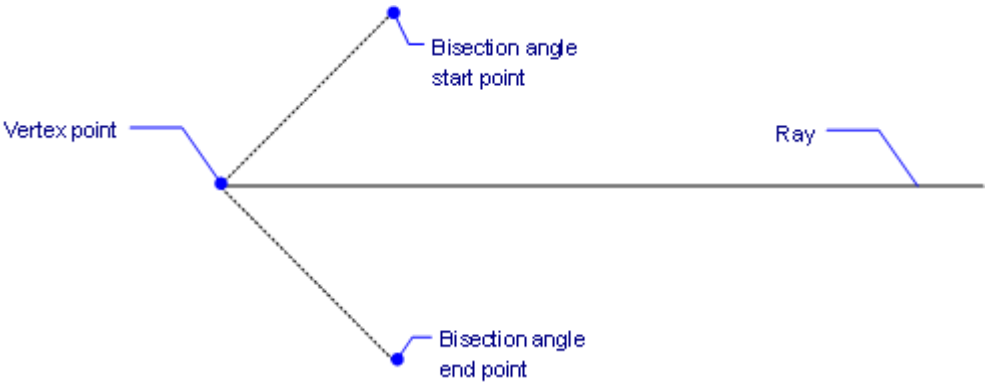
Direction: (Pick another point, or press Enter to exit command.)

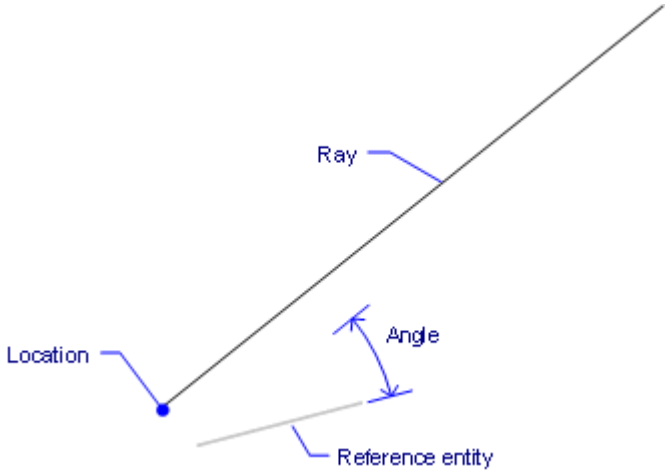
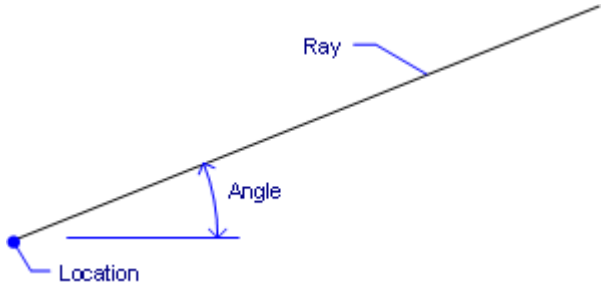
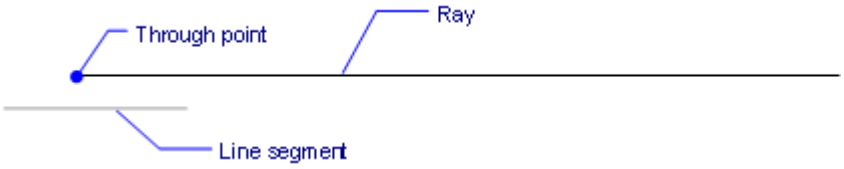


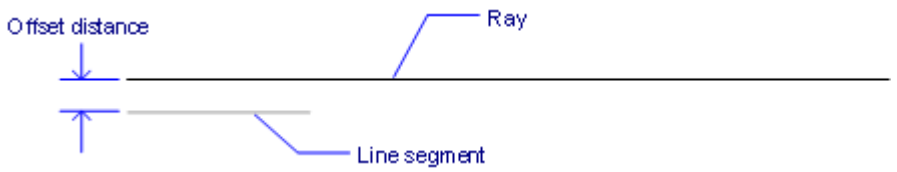
A ray has a starting point but no ending point; the other end stretches to infinity.

Command Options

Option	Description
Start of ray	Specifies the starting point of the ray. Pick a point in the drawing, or enter x,y coordinates; prompts you: Direction - specify the direction of the ray. The Direction prompt repeats so that you can draw multiple rays from the same starting point. Press Enter to exit the command.
Bisect	Draws a ray that bisects entities perpendicularly; prompts you: Entity/<Vertex point> - enter an option.
Entity	Draws rays perpendicular to the midpoint of lines and polyline segments, and the center point of arcs and polyarcs; prompts you: Select line,arc, or polyline segment to bisect - select a line, an arc, a polyline segment, or a polyarc. • Line or polyline segment - draws ray perpendicular to the segment's midpoint:  • Arc or polyarc - draws ray perpendicular to the arc's center point and through its midpoint:

	 When you select a line or polyline segment, BricsCAD prompts you: Select side for ray bisection - choose the side on which the ray should emerge. The 'Select line,arc, or polyline segment to bisect' prompt repeats so that you can draw rays on other entities. Press Enter to exit the command. This option works with splined polylines, but not with spline entities.
Vertex point	Draws rays that start at the vertex point and bisect two pick points; prompts you: Bisection angle start point - specify one part of the bisection angle. Bisection angle start point - specify the other part of the bisection angle. 
Horizontal	Draws rays horizontal to the current UCS; prompts you: Location - specify the horizontal location of the ray; pick a point in the drawing, or enter x,y coordinates. The 'Location' prompt repeats for drawing multiple horizontal rays at different starting points. Press Enter to exit the command.
Vertical	Draws rays vertical to the current UCS; prompts you: Location - specify the vertical location of the ray; pick a point in the drawing, or enter x,y coordinates. The 'Location' prompt repeats for drawing multiple vertical rays at different starting points. Press Enter to exit the command.
Angle	Draws rays at angles; prompts you: Reference/<Enter angle (0)> - enter an option.
Reference	Draws rays at an angle relative to another entity; prompts you: Select reference entity - choose a line. The line provides the reference (base) angle. Enter angle - specify the angle, measured relative to the entity.

	Location - specify the angle's start point; need not coincide with the entity's end points.  The 'Location' prompt repeats for drawing multiple rays at the same angle.
Enter angle	Specifies the angle; enter an angle or pick two points to indicate the angle; prompts you: Location - specify the starting point of the ray.  The 'Location' prompt repeats for drawing multiple rays at the same angle.
Parallel	Draws rays parallel to lines and polyline segments; prompts you: Through point/<Offset distance for parallel infinite line> (Through) - enter an option. This option works only with line segments.
Through point	Specifies the point through which to draw the ray; prompts you: Select entity for parallel infinite line - choose the entity to offset with a ray. Through the point - specify the point through which to draw the ray; this point also specifies the ray's start point. 
Offset distance for parallel infinite line	Specifies the offset distance. Enter a value or pick two points; prompts you: Select entity for parallel infinite line - choose the entity to offset with a ray. Side for parallel infinite line - specify the side on which to place the ray.

	 The ray's start point is parallel to the line's start point. The 'Select entity for parallel infinite line' prompt repeats for drawing multiple rays at different offset distances.
Direction	Specifies the direction of the ray. Pick a point, or enter x,y coordinates.

Procedures

Drawing rays

Grips Editing

Rays can be edited directly through grips:

1. Select the ray. Notice that it has two grips.
2. Drag the grips to perform the following actions:
    - Drag the endpoint grip to move the ray.
    - Drag the other grip to rotate the ray.

Related Commands

XLine - draws infinite lines.
Properties - modifies the properties of rays and xlines.

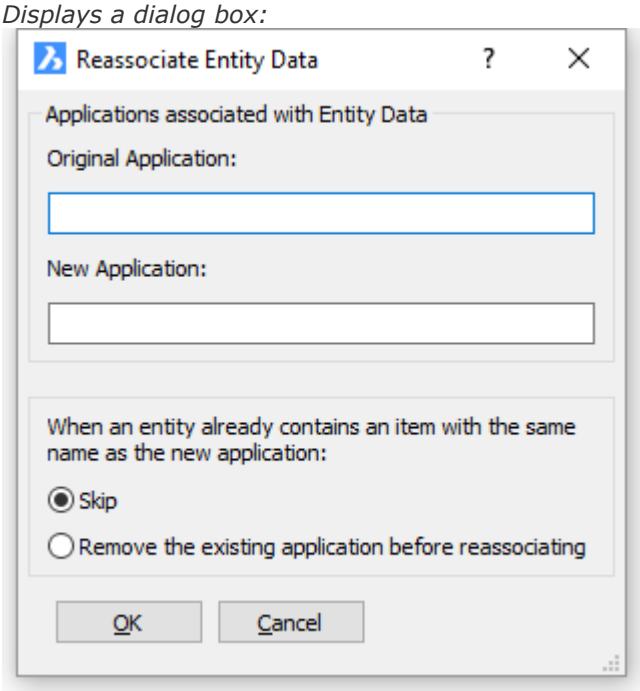
ReAssocApp

Associates extended entity data with applications (short for "reassociate application").

Accessing the Command

command bar: *reassocapp*
menu bar: *Tools | Entity Data | Reassociate Entity Data*
toolbar: *Entity Data |*

: ***reassocapp***
Displays a dialog box:



*Enter application names and then click **OK**.*

Command Options

Option	Description
Original Application	Specifies the name of the original application.
New Application	Specifies the name of the new application. Entities will be reassociated with this application name.
Skip	Ignores the change.
Remove the existing application before reassociating	Replaces the application name.

Related Commands

CopyEData - copies extended entity data to other entities.
DelEData - erases extended entity data from entities.
EditEData - creates and edits extended entity data.
MoveEData - moves extended entity data to other entities.

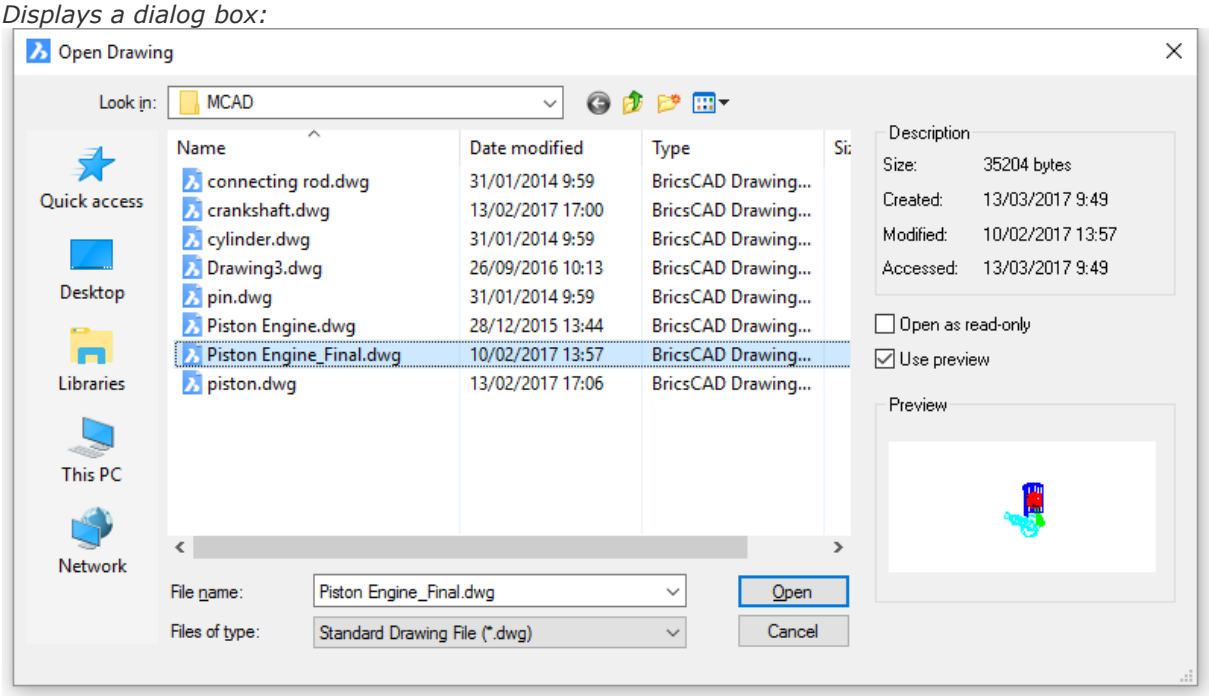
Recover

Recovers data from damaged DWG files.

Accessing the Command

command bar: *recover*
menu bar: *File | Recover*

: ***recover***
Displays a dialog box:



Choose a file, and then click **Open**.

If the drawing file contains damaged data, BricsCAD reports the changes it makes.

For example:

Name: Dwg file;

Value: CRC does not match in Object (167);

Name: AcDbPolygonMeshVertex(130);

Value: Owner Id (0);

Validation: Invalid;

Replaced by: Set to AcDbPolygonMesh(F0).

Command Options

No options.

Procedures

To analyze the integrity of the current drawing
To repair a drawing

Related Commands

Audit - checks drawings that are open.
ProxyInfo - displays a information about proxy entities, contained in the current drawing, in a dialog box.
RecoverAll - recovers a drawing with all nested XRef drawings, each drawing is saved with .BAK copy and .ADT report.

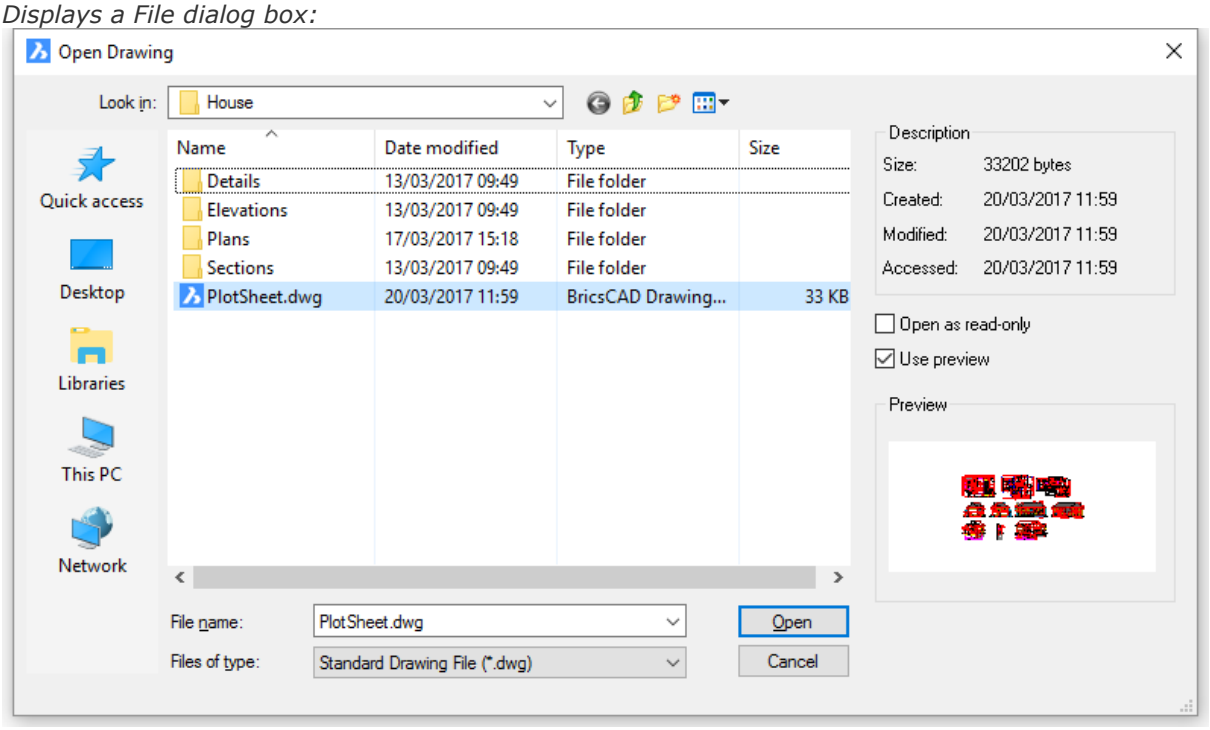
RecoverAll

Recovers a drawing with all nested XRef drawings, each drawing is saved with .BAK copy and .ADT report.

Accessing the Command

command bar: *recoverall*

: ***recoverall***
Displays a File dialog box:



Select the drawing, the click the *Open* button.
The selected drawing file and all attached xrefs, including all nested xrefs, are opened, repaired, saved and closed.
The report of the audit report of the repair is saved to a *.adt file.

Command Options

No command options.

Related Commands

Audit - repairs drawings that are open.
Recover - recovers data from damaged DWG files.

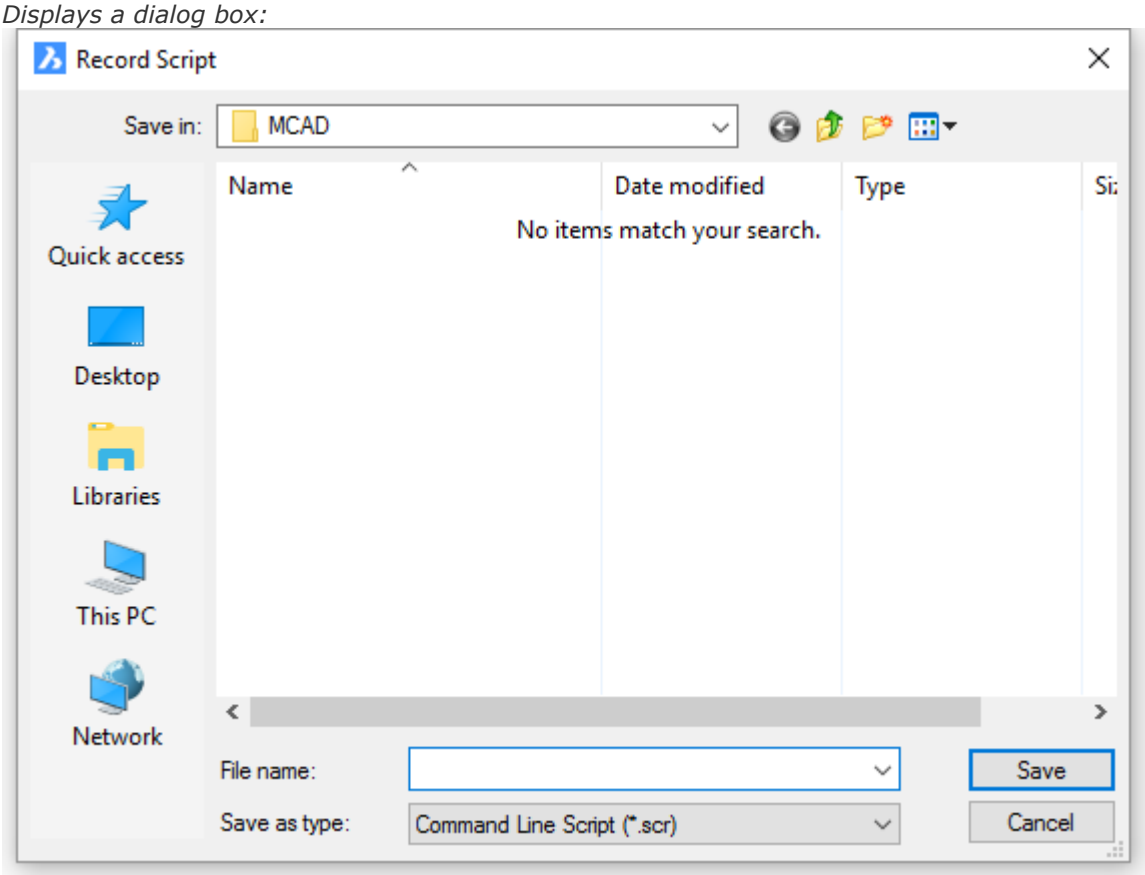
RecScript

Records keystrokes to an SCR file for playback with the Script command (short for "record script").

Accessing the Command

command bar: *recscript*
menu bar: *Tools | Record Script*
toolbar: *Tools |*

: ***recscript***
Displays a dialog box:



Specify a file name, and then click Save.
Enter drawing and editing commands, and pick points in the drawing.
When done, enter the StopScript command.

To playback scripts, use the **Script** command.

Command Options

Option	Description
File name	Specifies the name of the SCR script file.

Related Commands

StopScript - stops recording the script, and saves it to an SCR file.
Script - loads and runs a SCR script file.
RScript - reruns the script.
Delay - pauses the execution of the script for the specified number of milliseconds.
Resume - resumes a stopped script.

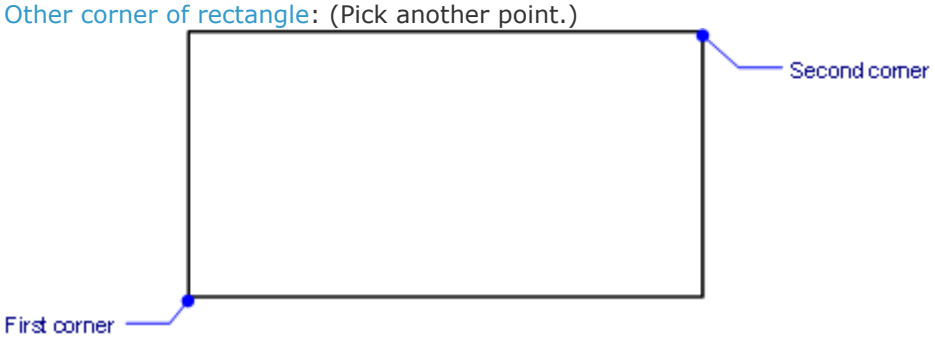
Rectang

Draws rectangles and squares from polylines (short for "rectangle").

Accessing the Command

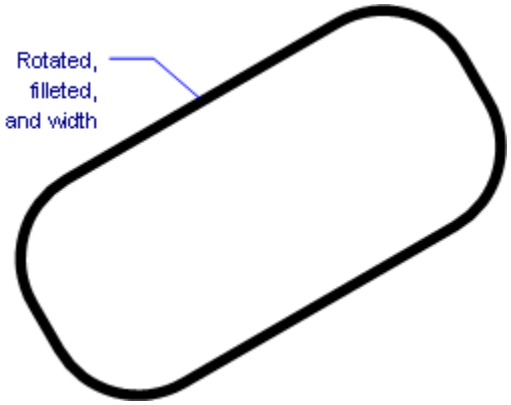
command bar: *rectang*
aliases: *rect, rectangle*
menu bar: *Draw | Rectangle*
toolbar: *Draw | Polygon |*

: ***rectang***
Prompts you in the command bar:
Select first corner of rectangle or [Chamfer/Fillet/Rotated/Square/Elevation/Thickness/Width of line/Area/Dimensions]: (Pick a point, or enter an option.)
Other corner of rectangle: (Pick another point.)



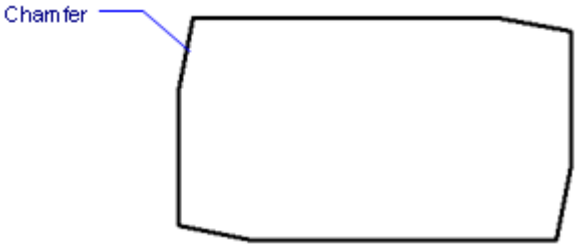
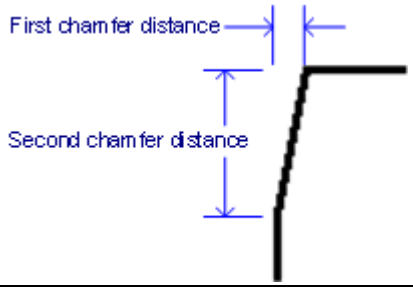
Rectangles are created as a closed polyline.

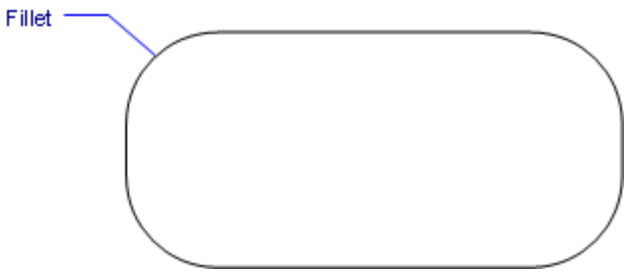
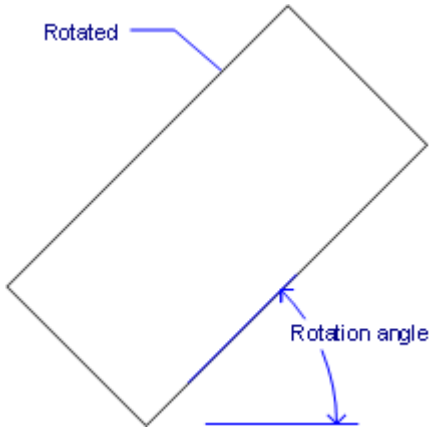
You can combine options, such as creating a rotated rectangle with width and filleted corners:

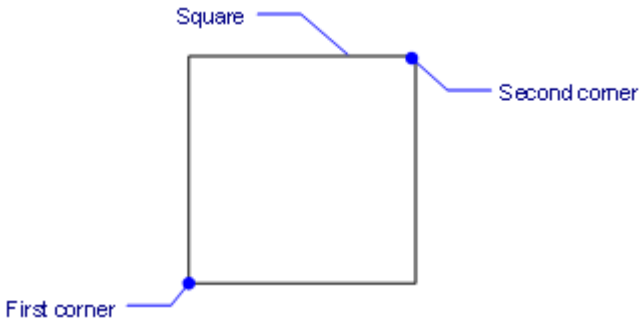
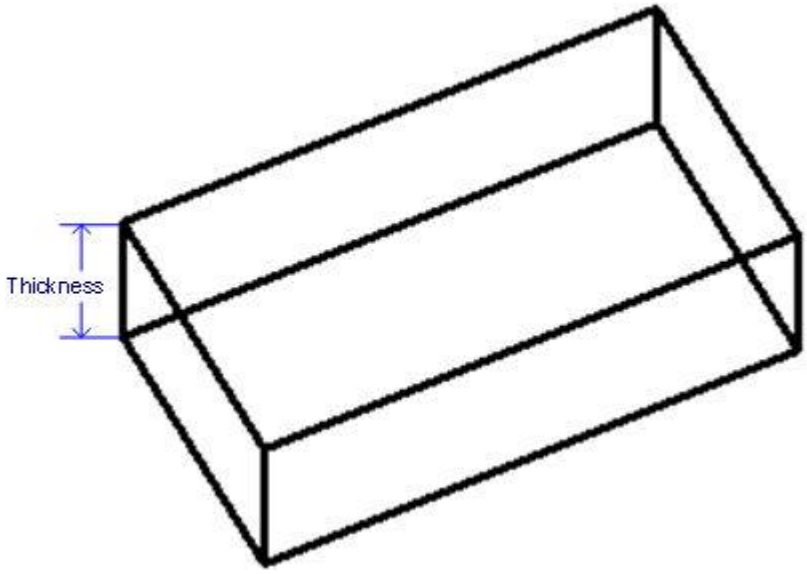


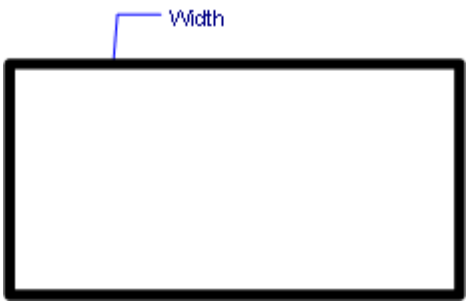
Command Options

Option	Description
Select first corner of rectangle	Specifies the first corner of the rectangle.
Other corner of rectangle	Specifies the opposite corner of the rectangle.
Chamfer	Applies a chamfer to all four corners of the rectangle. When chamfer distances = 0, prompts you:

	Defaults/First chamfer distance to use for all rectangles - enter an option. When chamfer distances are not 0, prompts you: Off/Defaults/First chamfer distance to use for all rectangles - enter an option.  The same chamfer is applied to all four corners. You can specify chamfer distances only; you cannot specify a chamfer angle. You can also edit the chamfers later with the Chamfer command.
Off	Turns off chamfering; uses 0 as the chamfer distances.
Defaults	Uses the default chamfer distances as stored by the Rectang command. This command does not use the ChamferA and ChamferB system variables.
First chamfer distance to use for all rectangles	Specifies the first chamfer distance; enter 0 or a positive number. 
Second chamfer distance to use for all rectangles	Specifies the other chamfer distance; enter 0 or a positive number.
Elevation	Specifies the elevation of the rectangle; prompts you: Elevation to use for all rectangles - specify the elevation; enter a positive or negative number. The elevation is the height above or below the x,y-plane. See Elev command.
Fillet	Applies a fillet radius to all four corners. When fillet radius = 0, prompts you: Fillet distance to use for all rectangles - specify the fillet radius; enter 0 or a positive number. When fillet radius not 0, prompts you: Off/Default/Fillet distance to use for rectangles - enter an option.

	 The same fillet is applied to all four corners. *Caution*: when the fillet radius is larger than the rectangle, an invisible rectangle is drawn. See Fillet command.
Off	Turns off filleting; uses 0 as the fillet radius.
Default	Uses the default fillet radius as stored by the Rectang command. This command does not use the FilletRad system variable.
Fillet distance to use for rectangles	Specifies the fillet radius; enter 0 or a positive number.
Rotated	Rotates the rectangle; delays the prompt until after the 'Other corner of rectangle' prompt: Rotation angle for rectangle - specify the rotation angle; pick a point or enter an angle.  Positive angles are measured counter-clockwise from the positive x axis. See Rotate command.
Square	Draws a square; delays the prompt until after the 'Select first corner of rectangle' prompt: Second corner of square - specify the opposite corner; pick a point, or enter x,y coordinates.

	 See Polygon command.
Thickness	Specifies the thickness of the rectangle. When thickness = 0, prompts you: Rectangle thickness - specify the thickness; enter 0 or a positive number. When fillet radius not 0, prompts you: Default/Rectangle thickness - enter an option.  The thickness is the height of the rectangle in the z direction; it is measured upwards from the elevation. See Elev command.
Default	Uses the default thickness as stored by the Rectang command. This command does not use the Thickness system variable.
Rectangle thickness	Specifies the thickness; enter 0 or a positive number.
Width	Specifies the width of the rectangle's lines; prompts you: Width to use for all rectangles - enter 0 or a positive number.

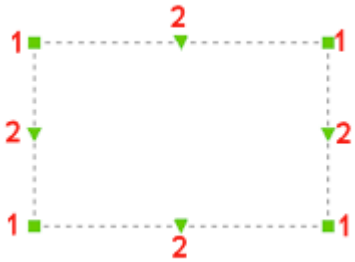
	 The width is applied equally to all four segments making up the rectangle. For tapered widths, edit the rectangle with the PEdit command.
Area	Specifies the area of the rectangle prompts you: Enter Area of rectangle <current value>: enter a non-zero value or press enter to accept the current value Calculate rectangle dimensions based on [Length/Width]<Length>: enter L or W Enter rectangle length <current value>: enter a non-zero value or Enter rectangle width <current value>: enter a non-zero value
Dimensions	Specifies the dimensions of the rectangle prompts you: Length to use for rectangles <current value>: enter a non-zero value or press enter to accept the current value or specify the length of the rectangle by clicking two points. Width to use for rectangles <current value>: enter a non-zero value or press enter to accept the current value or specify the width of the rectangle by clicking two points.

Procedures

Drawing rectangles

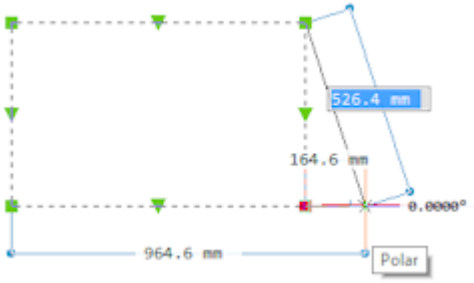
Grips Editing

Rectangles can be edited directly through grips:

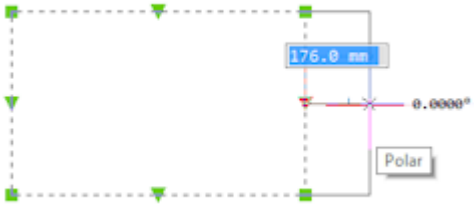


Select the rectangle. Notice that it has a square grip at each vertex (1) and a triangular grip at the midpoint of each side (2)*.
* If GRIPS = 1, midpoint grips do not display.
Press and hold the Shift key to select multiple vertices.
If *Dynamic Dimensions* (DYN) and *Polar Tracking* (POLAR) are active you can enter the distance(s) in the dynamic entry fields.

1. Drag a vertex grip (1) to stretch the a vertex.



2. Drag a midpoint grip (2) to stretch the side segment.



Related Commands

PLine - draws polyline segments and arcs.
3dPoly - draws polylines with x,y,z coordinates.
Polygon - draws regular polygons out of polylines.
Donut - draws wide circles from polylines.
PEdit - edits polylines.

Redefine

Activates commands that were deactivated with the Undefine command.

Accessing the Command

command bar: *redefine*

: ***redefine***
Prompts you in the command bar:
Command to redefine: (Enter name of command undefined with *Undefine* command.)

Command Options

Option	Description
Command to redefine	Specifies the name of the command to redefine.

Related Commands

Commands - lists the names of commands.
Customize - creates new commands through macros and LISP routines.
Undefine - removes commands from BricsCAD, temporarily.

Redo

Reverses the action of the last U or Undo command.

Accessing the Command

command bar: *redo*
shortcut: *Ctrl+Y*
menu bar: *Edit | Redo*
toolbar: *Standard |*

: ***redo***
Prompts you in the command bar:

Redo: (*commandName*)

This command works only when the **U** or **Undo** commands were used at least once in the current editing session.

When the U or Undo commands have not been used since the drawing was opened, BricsCAD responds:

There is nothing to redo.

Command Options

No options.

Related Commands

U - undoes the last command.
Undo - undoes one or more commands.
Oops - unerases the last erased object, including newly created blocks.
Zoom - utilizes the Previous option to show the previous view.

Redraw

Redraws the current viewport to clean it up.

Accessing the Command

command bar: *redraw*
transparent: *'redraw*
alias: *r*
menu bar: *View | Redraw*
toolbar: *View | Redraw/Regen |*

: ***redraw***

Displays no prompt in the command bar.

This command redraws the current viewport; use the RedrawAll command to redraw all viewports. This command removes blip marks left by the Blipmode command.

Command Options

No options.

Related Commands

Regen - regenerates the current viewport.
RegenAll - regenerates all viewports.
RedrawAll - redraws all viewports to clean them up.

RedrawAll

Redraws all viewports to clean them up.

Accessing the Command

command bar: *redrawall*
transparent: *'redrawall*
alias: *ra*
toolbar: *View | Redraw/Regen |*

: ***redrawall***

Displays no prompt in the command bar.

This command redraws all viewports; use the Redraw command to redraw the current viewport.

This command removes blip marks left by the Blipmode command.

Command Options

No options.

Related Commands

Regen - regenerates the current viewport.
RegenAll - regenerates all viewports.
Redraw - redraws the current viewport to clean it up.

RedSdkInfo

Displays rendering related hardware and driver specifications in the command bar.

Accessing the Command

command bar: *redsdkinfo*

: ***redsdkinfo***

Prints a report in the command window.
Press F2 to open the Prompt History window:

```
: redsdkinfo
RenderUsingHardware: Yes
Operating system: Win_seven64
Desktop composition (Aero) is enabled.
Local GPU:
    GPU information:
        name:     'Intel(R) HD Graphics Family'
        chipset:  'GMAHD'
        class:    'HW_INTEL_GMA_4000'
        asicID:   '278'
        vendorID: '32902'
        recommended driver: 'Intel(r) Graphics Media Accelerator Driver 15.21.10.64.2291'
    Installed driver:
        driver:      ''
        dll version: ''
        validation: 'NO_DRIVER_INSTALLED'
        You could try out this driver: 'http://www.redway3d.com/drivers/Win_seven64_Intel(r)
Graphics Media Accelerator Driver 15.21.10.64.2291'
Local GPU:
    GPU information:
        name:     'AMD Radeon HD 6800M Series'
        chipset:  'Manhattan'
        class:    'HW_ATI_R800'
        asicID:   '26792'
        vendorID: '4098'
        recommended driver: 'ATI Catalyst 11.1'
    Installed driver:
        driver:      ''
        dll version: '6.14.10.11013'
        validation: 'KNOWN_ERRORS'
        You could try out this driver: 'http://www.redway3d.com/drivers/Win_seven64_ATI
Catalyst 11.1'
```

Command Options

No Options

Related Commands

Render - generates a photorealistic rendering using materials and lights through a dialog box.
ShadeMode - sets the shading style for 3D entities.

RefClose

Closes the reference editor.

Accessing the Command

command bar: *refclose*
menu: *Tools | External Reference Editing | Reclose and Save* or *Refclose and Discard*
toolbar: *Ref Edit |* or

: ***refclose***
Prompts you in the command bar:

Enter option [Save/Discard reference changes] <Save>: *(Type S or D.)*

When you use this command on its own, BricsCAD complains:
** Command not allowed unless a reference is checked out with REFEDIT command **

Command Options

Option	Description
Save	Saves the changes back to the block reference.
Discard	Does not save the changes; the block is unchanged.

Related Commands

-RefEdit - edits block references and externally-referenced drawings through the command bar.
RefEdit - edits block references and externally-referenced drawings through a dialog box.

RefEdit

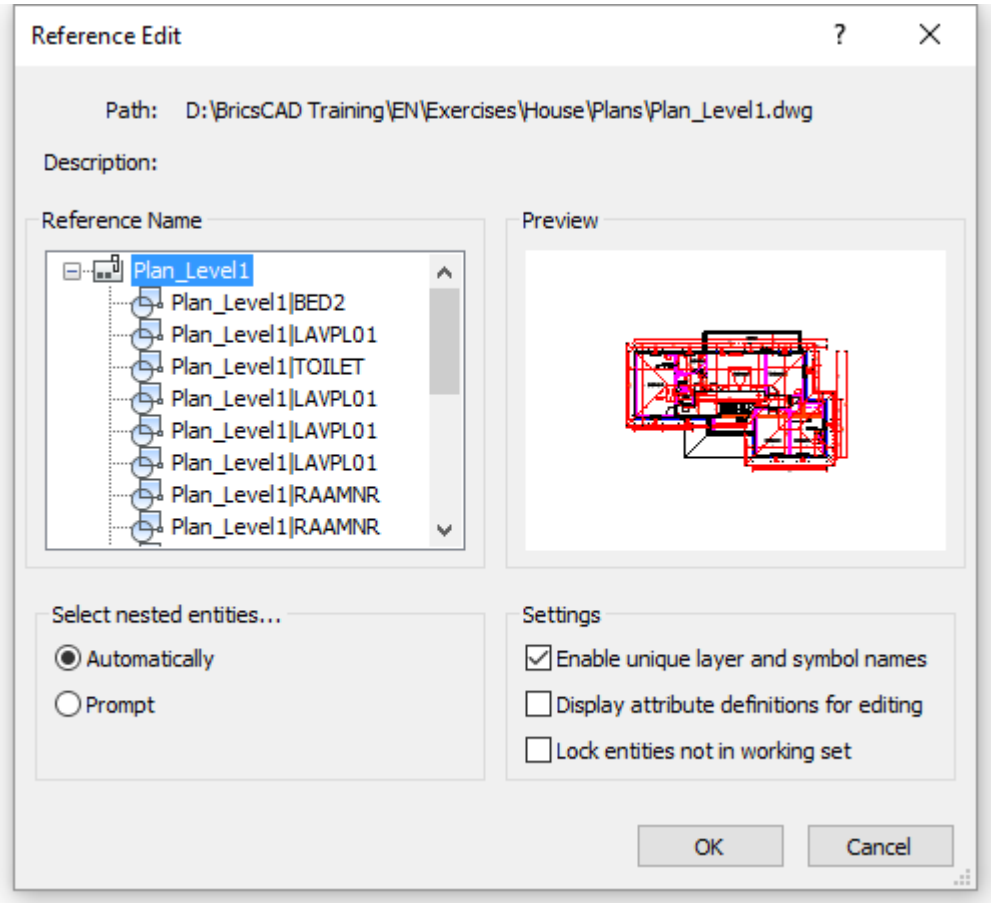
Edits block references and externally-referenced drawings through a dialog box (short for "reference editor").

Accessing the Command

command bar: *refedit*
transparent: *'refedit*
menu: *Tools | External Reference Editing | Refedit*
toolbar: *Ref Edit |*

: ***refedit***
Prompts you in the command bar:
Select reference: ***(Choose a block or an xref.)***

Displays a dialog box:



Choose options, and then click OK.

Prompts you in the command bar:
Use REFCLOSE or the Refedit toolbar to end reference editing session.

Make changes to the reference, and then use the **RefClose** command to save the changes.

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.

Command Options

Option	Description
Select reference	Chooses the reference to edit. Pick a single block or externally-referenced drawing. Only one reference can be edited at a time.
Dialog Box	
Reference Name	Lists the name of the block and nested blocks, if any.
Select nested entities	Determines how nested blocks/xrefs are handled.
Automatically	Nested entities are included automatically.
Prompt	Nested entities can be selected manually. After clicking OK to close this dialog box, you are prompted in the command bar: Select nested objects - select one or more nested entities. Select nested objects - select more nested entities, or press Enter to continue.
Settings	Determines how entities are treated during editing.
Enable unique layer and symbol names	Toggles the use of 0 in layer and other names: Yes - prefixes names with *filename0*. No - does not prefix names.
Display attribute definitions for editing	Toggles the inclusion of attributes: Yes - attribute definitions can be edited; attribute values are invisible. No - attributes are unavailable for editing. Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on.
Lock objects not in working set	Toggles locking of entities not being edited: Yes - locks entities outside the reference being edited. No - does not lock other entities. Locked entities can be seen but not edited.

Procedures

Editing blocks and external references

Related Commands

-RefEdit - edits references in-place through the command bar
Refclose - closes the reference editor
Refset - adds and removes entities from the reference being edited
XOpen - opens an externally-referenced drawing (Xref) for editing in a separate window

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
DdAtte - edits the values of attributes through a dialog box.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces blocks to constituent entities and attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits block and attribute values and properties.

-RefEdit

Edits block references and externally-referenced drawings through the command bar (short for "reference editor").

Accessing the Command

command bar: *-refedit*

: ***-refedit***
Prompts you in the command bar:

Select reference: **(Pick a reference.)**

Select nesting level [Ok/Next] <Next>: **(Type O or N.)**

Enter entity selection method [All/Nested] <All>: **(Type A or N.)**

Display attribute definitions [Yes/No] <No>: **(Type Y or N.)**

Use REFCLOSE or the Refedit toolbar to end reference editing session.

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.

Command Options

Option	Description
Select reference	Chooses the reference to edit. Pick a single block or externally-referenced drawing. Only one reference can be edited at a time.
Select nesting level	Choose the nested reference to edit: Ok - edits the entire block. Next - BricsCAD highlights the next nested level within the reference; repeats 'Select nesting level' prompt until you enter O.
Enter entity selection method	Determines how nested entities are selected: All - selects all nested entities automatically. Nested - prompts you to selects nested entities: Select nested objects - select one or more nested entities. Select nested objects - select more nested entities, or press Enter to continue.
Display attribute definitions	Toggles the inclusion of attributes: Yes - attribute definitions can be edited; attribute values are invisible. No - attributes are unavailable for editing. Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on.

Related Commands

RefEdit - edits references in-place through a dialog box.
Refclose - closes the reference editor
Refset - adds and removes entities from the reference being edited
XOpen - opens an externally-referenced drawing (Xref) for editing in a separate window

AttDef - defines attributes through a dialog box.
-AttDef - defines attributes through the command bar.
AttDisp - toggles the display of attribute text.
AttRedef - redefines a block and updates associated attributes.
AttSync - synchronizes attribute definitions in all block references of a specified block definition.
BattMan - manages the attributes of a block definition.
DdAtte - edits the values of attributes through a dialog box.
Block and -Block - create blocks from attribute definitions and entities.
EAttEdit - edits attribute values and most properties.
Explode - reduces blocks to constituent entities and attributes to tags.
Insert - inserts blocks and prompts for attribute values through a dialog box.
-Insert - insert blocks and prompts for attribute values through the command bar.
Properties - edits block and attribute values and properties.

RefSet

Adds and removes entities from the reference being edited.

Accessing the Command

command bar: *refset*
Tools | External Reference Editing | Add to Refedit or *Remove from Refedit*
toolbar: *Ref Edit |* *or*

: ***refset***
Prompts you in the command bar:

Transfer objects between the RefEdit working set and host drawing...

Enter an option [Add/Remove] <Add>: ***(Type A or R.)***

When you use this command on its own, BricsCAD complains:

** Command not allowed unless a reference is checked out with REFEDIT command **

Command Options

Option	Description
Add	Adds entities from the drawing to the reference being edited.
Remove	Removes entities from the reference being edited; removed entities are placed in the drawing. Entities not in the reference are shown in gray.

Regen

Regenerates the current viewport (short for "regenerate").

Accessing the Command

command bar: *regen*
transparent: *'redrawall*
alias: *re*
menu: *View | Regen*
toolbar: *View | Redraw/Regen |*

: ***regen***

Displays no prompt in the command bar.

This command regenerates the current viewport. Use the RegenAll command to regenerate all viewports.

This command reads the DWG file and reconstructs the entire drawing on the screen.

Command Options

No options.

Related Commands

RegenAll - regenerates all viewports.
Redraw - redraws the current viewport to clean it up.
RedrawAll - redraws all viewports to clean them up.

RegenAll

Regenerates all viewports (short for "regenerate all").

Accessing the Command

command bar: *regenall*
alias: *rea*
toolbar: *View | Redraw/Regen |*

: ***regenall***

Displays no prompt in the command bar.

This command reads the DWG file and reconstructs the drawing on the screen.

Command Options

No options.

Related Commands

Regen - regenerates the current viewport.
Redraw - redraws the current viewport to clean it up.
RedrawAll - redraws all viewports to clean them up.

RegenAuto

Controls when BricsCAD regenerates the drawing automatically.

Accessing the Command

command bar: *regenauto*
transparent: *'regenauto*

: ***regenauto***
Prompts you in the command bar:
REGENMODE is currently on: OFF/Toggle/<On>: *(Enter an option.)*

Command Options

Option	Description
OFF	Automatic regeneration is turned off. If a regeneration is required, BricsCAD does not execute it; prompts you: **Regen queued.** The regeneration takes place when you enter the **Regen** or **RegenAll** commands. When the regeneration is required immediately, prompts you: **About to regen--okay?** Answer yes or no.
Toggle	Toggles the state of the RegenMode system variable: • If on, turns it off. • If off, turns it on.
On	Turns on automatic regeneration. BricsCAD regenerates the drawing whenever it is required.

Region

Creates regions from 2D entities.

Accessing the Command

command bar: *region*
alias: *reg*
menu bar: *Draw | Region*
toolbar: *Draw |*

: ***region***

Prompts you in the command bar:

Select entities: (Choose one or more closed entities.)

Entities in set: 1

Select entities: (Choose more closed entities, or press Enter to continue.)

1 regions have been created.

Regions are ACIS objects, and can be used with solid editing commands, such as MassProp and Subtract.

The source entities are erased.

Command Options

Option	Description
Select entities	Specifies the entities to convert to regions. Choose one or more closed entities. Each entity is converted into a separate region. Closed entities include circles, closed polylines, and ellipses.

Related Commands

Extrude - extrudes 2D entities into 3D solid models.
Intersect - removes areas not in common between two or more regions.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Subtract - removes one region from a second.
Union - joins two or more regions into a single entity.

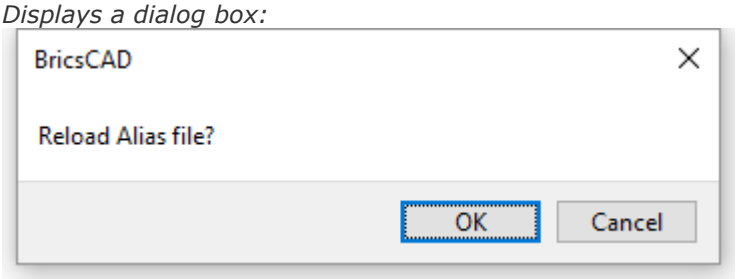
ReInit

Reloads the PGP alias file (short for "reinitialize").

Accessing the Command

command bar: *reinit*
alias: *ri*

: ***reinit***
Displays a dialog box:



Click OK or Cancel.

This command is used to load the PGP file after it has been edited externally.
The PGP file defines command aliases. See the "Customizing BricsCAD" ebook for details on aliases.

Command Options

Option	Description
OK	Reloads the PGP file into BricsCAD.
Cancel	Does not reload the file.

Related Command

Customize - creates and edits aliases.

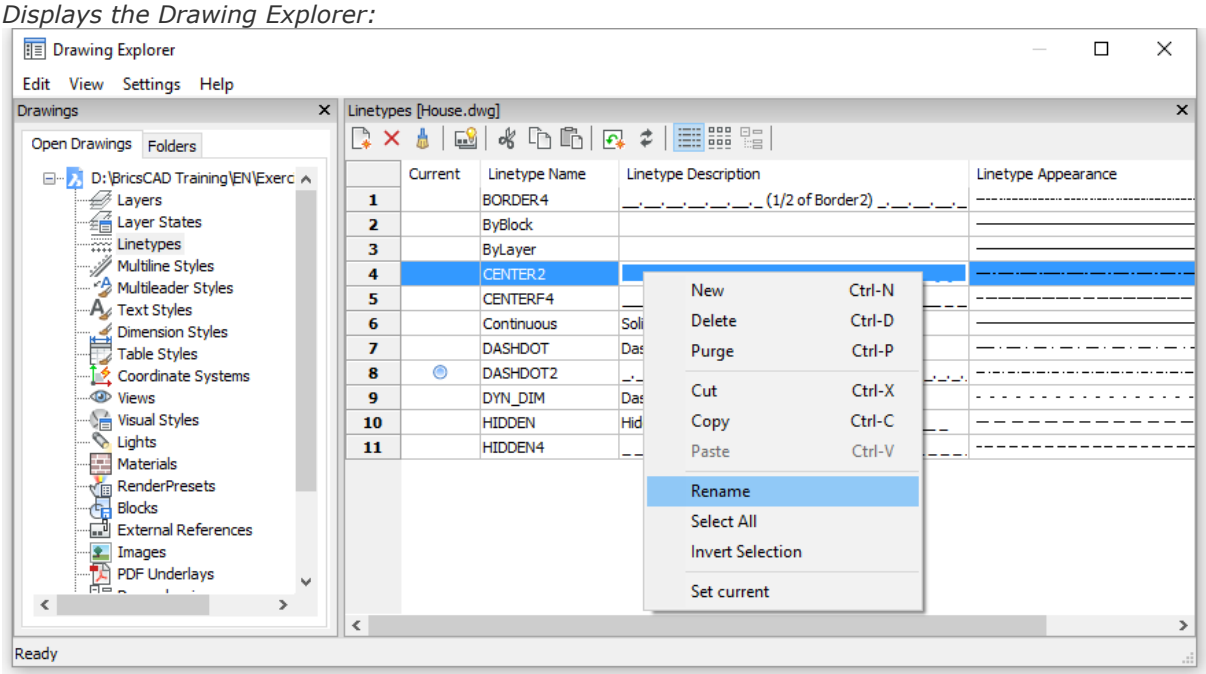
Rename

Renames named entities through the Drawing Explorer.

Accessing the Command

command bar: *rename*
aliases: *ren, ddrename*

: ***rename***
Displays the Drawing Explorer:



To rename an item, click twice on its name (do not double-click), then enter the new name.

Alternatively, right-click the item, then choose **Rename** in the shortcut menu.

You can change the names of most layers, layer states, most linetypes, UCSs, views, blocks, most text styles, and most dimension styles.

You cannot change the names of attached images and external references.

You cannot change the following names:

- Layers - 0
- Linetypes - ByBlock, ByLayer, Continuous
- Text styles - Standard
- Dimension styles - Standard

To rename at the command line, use the **-Rename** command.

Command Options

No options.

Related Command

-Rename - renames commands through the command bar.

-Rename

Renames named entities in the command bar.

Accessing the Command

command bar: *-rename*
alias: *-ren*

: ***-rename***
Prompts you in the command bar:

Rename [Block/Dimension style/LAyer/LineType/text Style/Table style/Ucs/VIew/ViewPort]: *(Enter an option.)*

To rename in the Drawing Explorer, use the Rename command.

Command Options

Option	Description
Block	Renames blocks; prompts you: Block to be renamed - enter the name of an existing block. New name for the Block name - enter the new name. If you enter the name of an existing block, BricsCAD prompts: **A Block with that name already exists.** New name for the Block name - enter a different name. See Block command.
Dimension style	Renames dimension styles; prompts you: Dimension style to be renamed - enter the name of an existing dimension style. New name for the Dimension style name - enter the new name. If you enter the name of an existing dimension style, BricsCAD prompts: **A Dimension style with that name already exists.** New name for the Dimension style name - enter a different name. See DimStyle command.
LAyer	Renames layers; prompts you: Layer to be renamed - enter the name of an existing layer. New name for the Layer name - enter the new name. If you enter the name of an existing layer, BricsCAD prompts: **A Layer with that name already exists.** New name for the Layer name - enter a different name. If you attempt to rename layer 0, BricsCAD prompts: **That name is reserved by BricsCAD, and cannot be renamed.** See Layer command.
LineType	Renames linetypes; prompts you: Linetype to be renamed - enter the name of an existing linetype. New name for the Linetype name - enter the new name. If you enter the name of an existing linetype, BricsCAD prompts: **A Linetype with that name already exists.**

	New name for the Linetype name - enter a different name. If you attempt to rename linetypes ByLayer, ByBlock, or Continuous, BricsCAD prompts: **That name is reserved by BricsCAD, and cannot be renamed.** See Linetype command.
text Style	Renames text styles; prompts you: Text style to be renamed - enter the name of an existing style. New name for the Text style name - enter the new name. If you enter the name of an existing style, BricsCAD prompts: **A Text style with that name already exists.** New name for the Text style name - enter a different name. See Style command.
Tablestyle	Renames table styles; prompts you: Table style to be renamed - enter the name of an existing style. New name for the Table style name - enter the new name. If you enter the name of an existing style, BricsCAD prompts: **A Table style with that name already exists.** New name for the Table style name - enter a different name.
Ucs	Renames user-defined coordinate systems; prompts you: UCS to be renamed - enter the name of an existing UCS. New name for the UCS name - enter the new name. If you enter the name of an existing UCS, BricsCAD prompts: **A UCS with that name already exists.** New name for the UCS name - enter a different name. See UCS command.
VIew	Renames views; prompts you: View to be renamed - enter the name of an existing view. New name for the View name - enter the new name. If you enter the name of an existing view, BricsCAD prompts: **A View with that name already exists.** New name for the View name - enter a different name. See View command.
VPort	Renames viewport configurations; prompts you: Viewport to be renamed - enter the name of an existing viewport configuration. New name for the Viewport name - enter the new name. If you enter the name of an existing viewport, BricsCAD prompts: **A Viewport with that name already exists.** New name for the Viewport name - enter a different name. See VPorts command.

Related Command

Rename - renames commands through the Drawing Explorer.

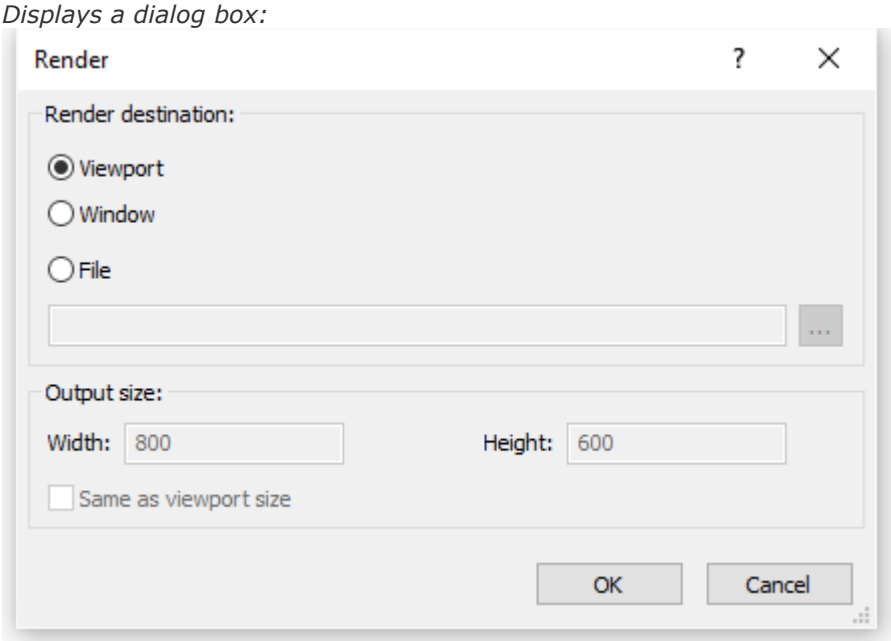
Render

Generates a photorealistic rendering using materials and lights through a dialog box.

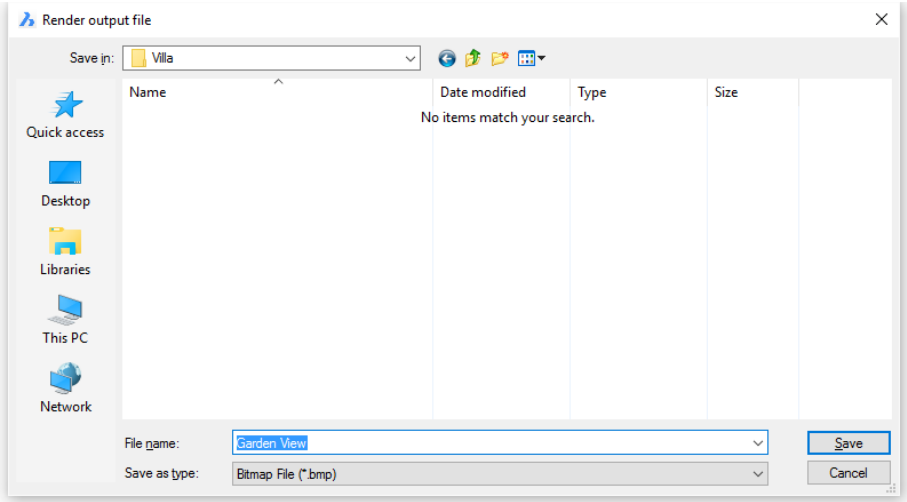
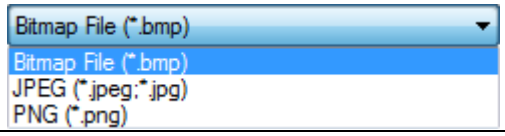
Accessing the Command

command bar: *render*
menu bar: *View|Rendering - Render*
toolbar: *Render|*

: ***render***
Displays a dialog box:



Command Options

Option	Description
Viewport	The current viewport is rendered.
Window	The rendered view shows in a separate window.
File	The rendered image of the current viewport is exported to a file. Click the browse button to open the *Render output file* dialog.  • Select an output folder. • Type a name in the File name field. • Select an image file type: 
Output Size	The *Width* and *Height* fields define the quality of the output image. The *Same as viewport size* option matches the ouput size or the size of the output window to the current viewport size.
OK	Starts calculating the rendering. The rendered image is calculated according to the current render preset settings (see the RenderPresets command). The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5. Note: high anti-alias values incur a high calculation cost.

Procedures

To render a viewport
To render in a window
To save a rendering to a file

Related Commands

AniPath - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
-Render - generates a photorealistic rendering using materials and lights through the command bar.
RenderPresets - allows to create and edit render presets, and to set the current render preset through a dialog box.

Background - sets the model space background for the current viewport and saved views.
Distantlight - creates a distant light.
Hide - removes hidden lines from 3D entities.
Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Materials - creates materials and edits their properties through the Drawing Explorer.
Pointlight - creates a point light.
RedSdkInfo - displays rendering related hardware and driver specifications in the command bar.
Shade - generates shaded images of 3D drawings.
ShadeMode - sets the shading style for 3D entities.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.
Weblight - creates a web light.

-Render

Generates a photorealistic rendering using materials and lights through the command bar.

Accessing the Command

command bar: *-render*

: ***-render***
Prompts you in the command bar:

Specify render preset [Draft/Low/Medium/High/Presentation/Other] <Medium>: (Specify a render preset or press Enter to accept the current preset.)

Specify render destination [Viewport/File/Render window] <Viewport>: (Choose an option or press Enter to accept the default.)

Command Options

Option	Description
Render preset	Sets the settings to be used for the rendering. See the RenderPresets command. Do one of the following: • Select one of the hard-coded presets (Draft, Low, High, Presentation) • Choose *Other*, to choose a user-defined preset. Prompts you: Enter custom render preset name or [?]: Type the name of a preset or type *?* to list the available custom presets.
Viewport	Renders the current viewport.
File	Saves the render output to a *.bmp file. Prompts you: Specify output file name: Type the full path and filename. Enter output width or [Viewport size] <xxx>: Do one of the following: • Press Enter to accept the default width. • Type the width of the image. • Type *V*, then press Enter to set the width and height to the size of the current viewport. Enter output height <600>: Type the height of the image or press Enter to accept the default height.
Render window	Displays the rendering in a separate window. Prompts you: Enter output width or [Viewport size] <800>: Do one of the following: • Press Enter to accept the default width. • Type the width of the image. • Type *V*, then press Enter to set the width and height to the size of the current viewport. Enter output height <600>: Type the height of the image or press Enter to accept the default height. Save also to file?[Yes/No] <No>: • Yes: prompts for an output file name.

	• No: displays the rendering in a separate window.
Specify render destination	Sets the output location for the rendering. The options are: • **Viewport -** the current viewport is rendered. • **File -** exports the rendering to an file. Prompts you: **Specify output file name:** type a file name. • Render view - Displays the rendering in a pop-up window. Prompts you: Enter output width or [Viewport size] <800> - type a value or choose *Viewport Size* to set the size of the pop-up window equal to the current viewport size. Enter output height <600> - type a value Save also to file?[Yes/No] <No> - If Yes, you are prompted: Specify output file name - Type a file name.

The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.
Note: high anti-alias values incur a high calculation cost.

Related Commands

AniPath - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
Render - generates a photorealistic rendering using materials and lights through a dialog box.
RenderPresets - allows to create and edit render presets, and to set the current render preset through a dialog box.

Background - sets the model space background for the current viewport and saved views.
Distantlight - creates a distant light.
Hide - removes hidden lines from 3D entities.
Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Materials - creates materials and edits their properties through the Drawing Explorer.
Pointlight - creates a point light.
RedSdkInfo - displays rendering related hardware and driver specifications in the command bar.
Shade - generates shaded images of 3D drawings.
ShadeMode - sets the shading style for 3D entities.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.
Weblight - creates a web light.

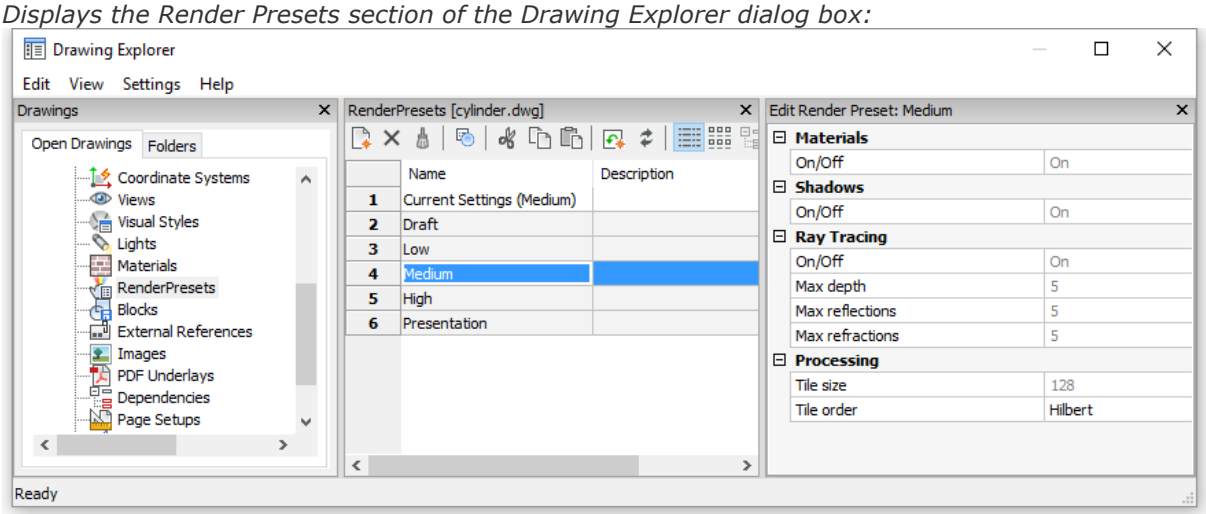
RenderPresets

Allows to create and edit render presets, and to set the current render preset through a dialog box.

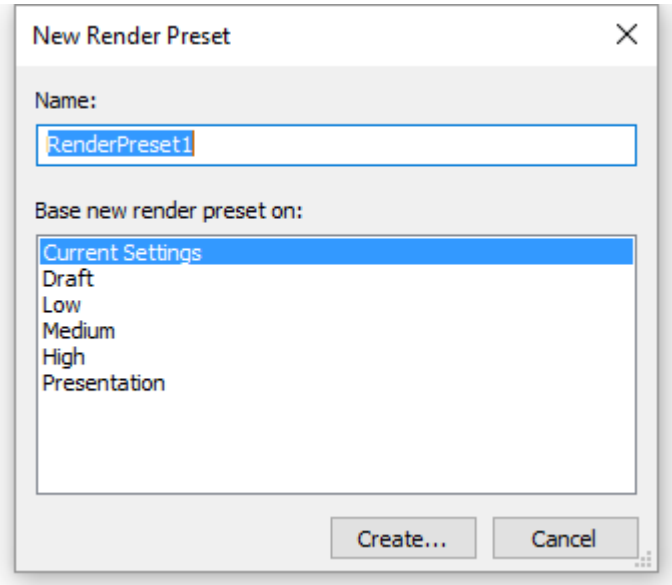
Accessing the Command

command bar: *renderpresets*
menu bar: *Tools | Drawing Explorer | Render Presets*

: ***renderpresets***
Displays the Render Presets section of the Drawing Explorer dialog box:



Command Options

Option	Description
	Creates a new render preset definition. 1. Click the *New* tool button ( ). The *Drawing Explorer - Render Presets* closes temporarily. The *New Render Preset* dialog box displays.  2. Type a name in the *Name* field. 3. Select an existing render preset which you want to use as a template for the new render preset. 4. Click the Create... button. The *Drawing Explorer - Render Presets* dialog box reopens. 5. (option) Type a description in the *Description* field (Detail View only).
	Deletes the currently selected user defined render preset(s). It is not possible to delete predefined render presets: *Draft*, *Low*, *Medium*, *High* and *Presentation*.
	Sets the selected render preset current. The current render preset is applied when using the Render command or when printing using the shade plot type *Rendered* as defined in *Shade Plot* setting in the Page Setup or the Print dialog. The name of the current render preset displays between brackets: e.g. Current Settings (Medium)? Optionally overrides can be defined in the the editor pane. If overrides exist, an asterisk displays in front of the current render preset name: e.g. Current Settings (*Medium)
	Displays the *Detail View* of the render presets.
	Displays the *Icon View* of the render presets.

Settings	
It is not possible to edit predefined render presets: *Draft*, *Low*, *Medium*, *High* and *Presentation*, except for *Processing > Tile order.*	
Materials	If *On*, uses material definitions for rendering. See Materials property of a layer or entity.
Shadows	If *On*, calculates shadows.
Ray Tracing	If *On*, calculates ray tracing. *Max. depth*, *Max refections* and *Max refractions* define the number of iterations for ray tracing calculations.
Processing	Tile size: sets the size of the tiles the rendered image is processed. Tile order: When you render to window the rendered image is generated tile by tile. This setting will define the order in which the tiles are generated. The options are: • Hilbert: This is the default setting, following a Hilbert curve: a continuous fractal space-filling curve first described by the German mathematician David Hilbert in 1891. See http://en.wikipedia.org/wiki/Hilbert_curve for a detailed description. • Spiral: Following a counterclockwise spiral built up from the center of the image • Left to right: In vertical columns starting from the bottom left corner of the image. • Right to left: In vertical columns starting from the bottom right corner of the image. • Top to bottom: In horizontal rows starting at the top left corner of the image. • Bottom to top: In horizontal rows starting at the bottom left corner of the image.

Related Commands

Render - generates a photorealistic rendering using materials and lights through a dialog box.
-Render - generates a photorealistic rendering using materials and lights through the command bar.

Explorer - Opens the Drawing Explorer dialog box.

ResetBlock

Resets dynamic blocks to default values.

Accessing the Command

command bar: *resetblock*

: ***resetblock***

Prompts you in the command bar:

Select entities: (Select dynamic blocks.)

Entities in set: 1

Select entities: (Select more dynamic blocks or press Enter.)

Command Options

No options.

Related Commands

-Insert - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.
Insert - inserts blocks, through a dialog box and prompts for attribute values through the command bar or a dialog box.

Resume

Resumes paused scripts.

Accessing the Command

command bar: *resume*
transparent: *'resume*

: ***resume***

Displays no prompts in the command bar.

This command resumes scripts after they have been stopped with Esc.

Run scripts with the Script command.

Command Options

No options.

Related Commands

RecScript - records scripts to SCR files.
StopScript - stops recording the script, and saves it to an SCR file.
Script - loads and runs a SCR script file.
RScript - reruns the script.
Delay - pauses the execution of the script for the specified number of milliseconds.

RevCloud

Draws revision clouds commonly used for redlining drawings (short for "revision cloud").

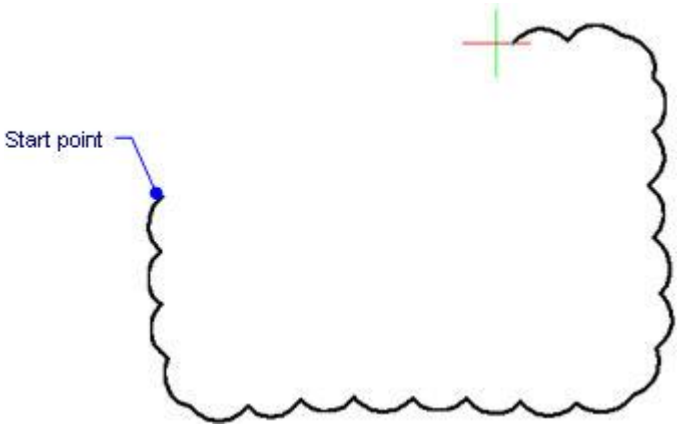
Accessing the Command

command bar: *revcloud*
menu bar: *Draw | Revision Cloud*
toolbar: *Draw | Polygon |*

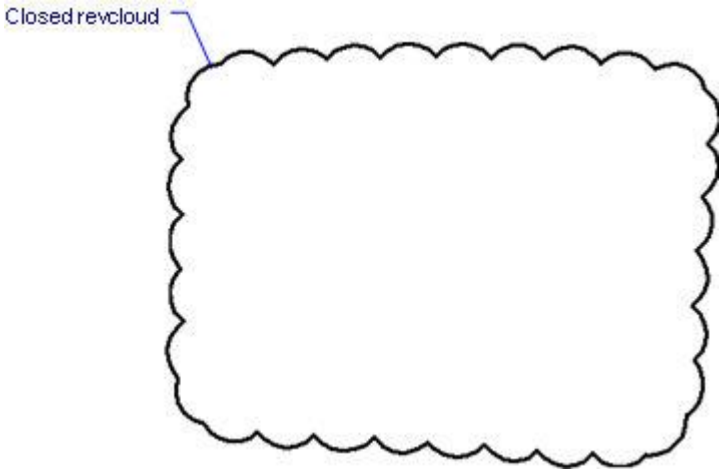
: ***revcloud***
Prompts you in the command bar:

Specify start point or [Arc length/Entity/Style] <Entity>: (Pick a point, or enter an option.)

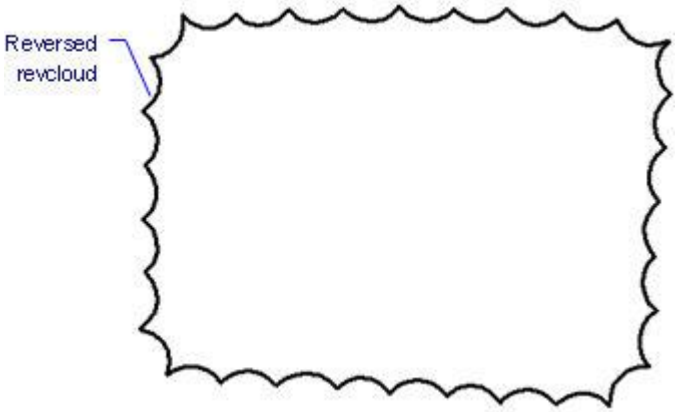
Guide crosshairs along cloud path... (Move the cursor to create the free form cloud shape.)



Revision cloud finished. (When the cursor reaches the starting point, the cloud closes automatically.)



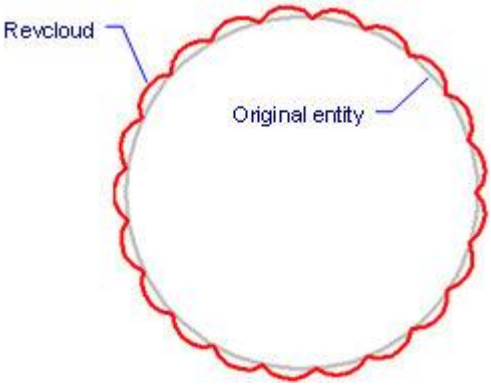
Reverse direction [Yes/No] <No>: (Type Y or N.)

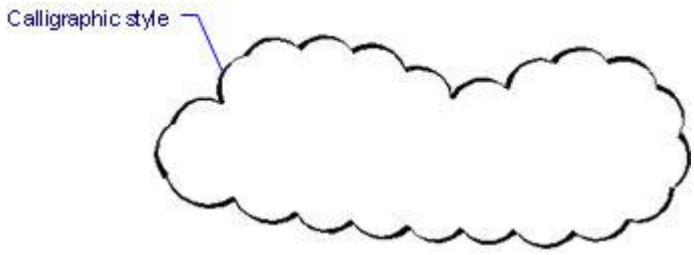


For red revision clouds, set the layer color to red.

A revision cloud is drawn as a closed polyline composed of arcs.

Command Options

Option	Description
Specify start point	The revision cloud closes automatically when the cursor reaches the start point.
Arc length	Specifies the shortest length for the arcs that form the cloud; prompts you: Specify minimum length of arc - enter a value.
Entity	Converts entities into a revision cloud; prompts you: Select entity - choose the entity to be converted into a revision cloud.  Entities can be open or closed. The original entity is erased.
Style	Toggles between normal and calligraphic drawing style; prompts you: Select arc style [Normal/Calligraphy] <Normal> - type N or C.
Normal	Draws normal arcs.
Calligraphy	Draws in a calligraphic style (made of tapered polyarcs).

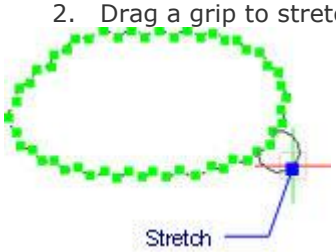
Procedures

Drawing revision clouds

Grips Editing

Revision clouds can be edited directly through grips:

1. Select the revcloud. Notice that it has many grips.
2. Drag a grip to stretch the cloud.



Related Commands

PEdit - edits polylines.
Sketch - draws freehand sketches.

Revolve

Revolves open or closed 2D entities and solid edges and faces to turn them into 3D solids or surfaces.

Accessing the Command

command bar: *revolve*
alias: *rev*
menu bar: *Model | 3D Solids | Revolve*
toolbar: *3D Solids |*

: ***revolve***
Prompts you in the command bar:

Select entities/subentities to revolve or set [MOde]: (Choose one or more closed 2D entities or set the mode.)

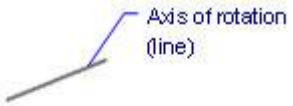
Entities in set: 1

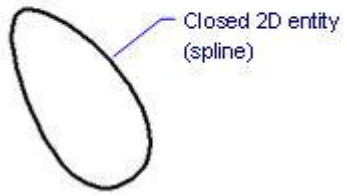
Select entities/subentities to revolve or set [MOde]: (Choose more closed 2D entities, or press Enter to continue.)

Specify start point for axis of revolution or define axis by [Object/Xaxis/Yaxis]: (Pick a point or enter an option.)

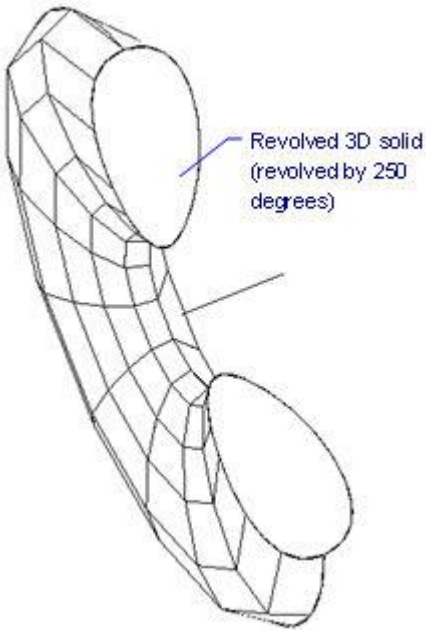
Specify endpoint of axis: (Pick a point.)

Specify angle of revolution <360>: (Press Enter to accept a 360° revolution or type a value.)





Specify angle of revolution <360>: (Enter the number of degrees, pick two points, or press Enter for the default.)



The axis can be defined by two points; a line or a polyline segment; or the x or y axis of the current coordinate system.

The axis must be outside of the entity being revolved.

Depending on the value of the DELOBJ system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities.

Command Options

Option	Description
Mode	Sets the type of entity being created. Prompts you: **Choose type of created entity [SOlid/SUrface] <SOlid>:** • Solid - Creates 3D solids. • Surface - Creates a 3D surface
Select entities/subentities	Specifies the entities and/or subentities to be revolved. Choose one or more open or closed 2D entities or solid edges or faces when in Surface mode. Choose one or more closed 2D entities or solid faces when in Solid mode.
Object	Specifies the axis of rotation by an object. Pick a line or a polyline segment.
Xaxis	Uses the x axis of the current coordinate system as the axis of rotation.
Yaxis	Uses the y of the current coordinate system axis as the axis of rotation.

Related Commands

dmRevolve - creates 3D solids by revolution of closed 2D entities or regions about an axis.

Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.
Union - joins two or more solid models into a single entity.

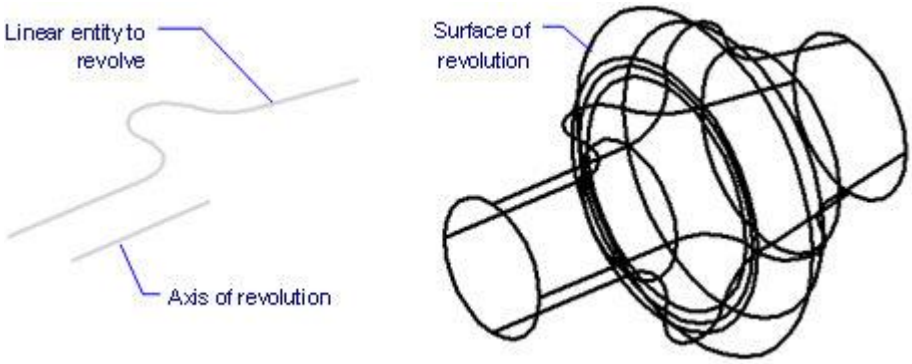
RevSurf

Creates a 3D mesh surface by revolving a linear entity around a line.

Accessing the Command

command bar: *revsurf*
menu bar: *Model | Meshes | Revolved Surface*
toolbar: *Meshes | 3D Surfaces |*

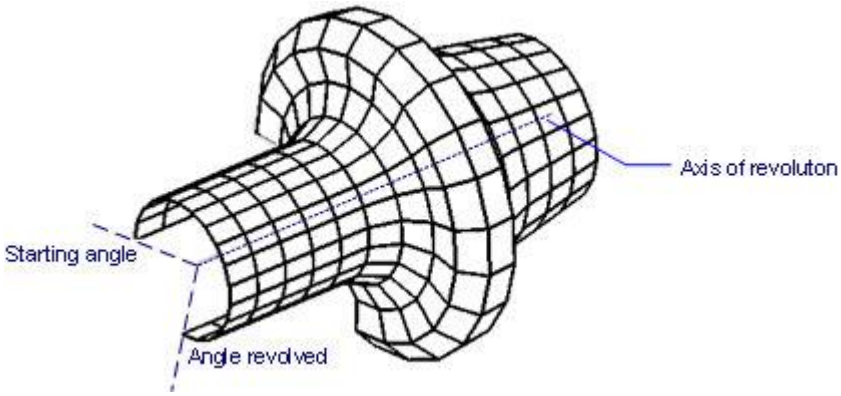
: ***revsurf***
Prompts you in the command bar:
Select a linear entity to revolve: (Pick a single entity.)
Select the axis of revolution: (Pick a line.)
Angle to begin surface of revolution <0>: (Specify an angle, or press Enter to accept the default.)
Degrees to revolve entity (+ for ccw, - for cw <360>): (Specify an angle with an optional negative sign, or press Enter to accept the default.)



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.

Command Options

Option	Description
Select a linear entity to revolve	Specifies a single entity to be revolved into a 3D surface mesh. You may choose a line, circle, arc, polyline, or spline; 3D objects do not work.
Select the axis of revolution	Specifies the entity about which the surface is revolved; you may choose a line or polyline; curved objects do not work.
Angle to begin surface of revolution	Specifies the starting angle of the revolution; the default is 0 degrees.
Degrees to revolve entity	Specifies the extent of the revolution; for a closed object, start with 0 and end with 360 degrees. The revolved surface is drawn counterclockwise; prefix the angle with a negative sign to draw the revolution clockwise. Note that this is not the ending angle, but the number of degrees by which the entity is revolved.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
Revolve - revolves open or closed 2D entities and solid edges and faces to turn them into 3D solids or surfaces.
Sweep - creates a 3D solid by sweeping a closed 2D entity along a path.
TabSurf - draws tabulated surfaces as mesh surfaces.

Ribbon

Displays the ribbon.

Accessing the Command

command bar: *ribbon*

: ***ribbon***

Command Options

No options.

Procedures

Exploring the ribbon
Creating a ribbon tab
Editing a ribbon tab
Creating a ribbon panel
Editing a ribbon panel
Editing a panel row

Related Commands

CleanScreenOff - displays user interface elements that were hidden by te CleanScreenOn command.

CleanScreenOn - hides elements of the user interface.
RibbonClose - hides the ribbon.
ToolPalettes - opens the Tool Palettes bar.
Customize - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, aliases, and shell commands.

RibbonClose

Hides the ribbon.

Accessing the Command

command bar: *ribbonclose*

: ***ribbonclose***

Command Options

No options.

Related Commands

CleanScreenOff - displays user interface elements that were hidden by te CleanScreenOn command.

CleanScreenOn - hides elements of the user interface.
Ribbon - displays the ribbon.

Rotate

Rotates entities in 2D.

Accessing the Command

command bar: *rotate*
alias: *ro*
menu bar: *Modify | 2D Rotate*
toolbar: *Modify |*

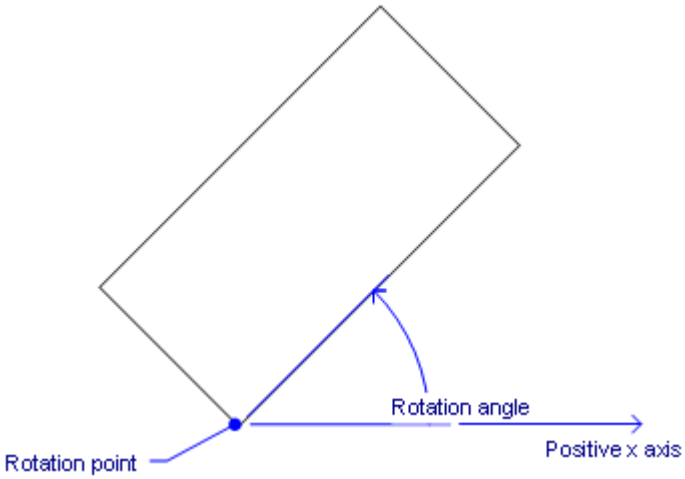
: ***rotate***
Prompts you in the command bar:

Select entities to rotate: (Select one or more entities.)
Entities in set: 1

Select entities to rotate: (Select more entities, or press Enter to continue.)

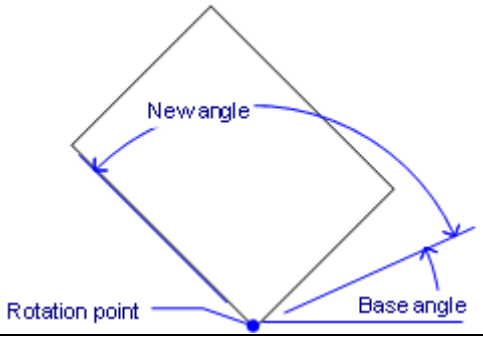
Rotation point: (Pick a point or enter x,y coordinates.)

Rotation angle or [Base angle/Copy] <0>: (Enter an angle, or choose an option.)



Command Options

Option	Description
Select entities to rotate	Chooses the entities to be rotated. You can choose one or more entities using any entity selection method. Press Enter to end entity selection.
Rotation point	Specifies the point about which the entities will rotate. Pick a point, or enter x,y coordinates. Use entity snaps for an accurate rotation point.
Rotation angle	Specifies the angle by which to rotate the entities: • Enter a point to show the angle. • Enter a positive or negative number to specify the angle. Positive numbers rotate entities counterclockwise from the positive x axis.
Copy	Creates a rotated copy of the selection set.
Base angle	Specifies the a new base angle; prompts you:

	Base angle - specifies the base angle. Do one of the following: • Type a value. • Pick the rotation point again. You are prompted: Second point - pick a point. The selected entities rotate around the rotation point and the dynamic rotation angle is controlled by the rotation point and the second point. New angle - specifies the angle by which to rotate the entities. Do one of the following: • Type a value. • Pick a point. This command normally rotates entities relative to the positive x axis. This option lets you define a different angle as the starting point. 

Procedures

Rotating a selection set
Rotating a selection set in reference to a base angle
To rotate entities using grips
To rotate the display in a viewport

Related Commands

Rotate3D - rotates entities about axes in 3D space.
Align - aligns one entity with another.
Snap - rotates the cursor, ortho angle, and grid.
UCS - rotates the coordinate system.

Rotate3D

Rotates entities about an axis in 3D space.

Accessing the Command

command bar: *rotate3d*
alias: *3drotate*
menu bar: *Modify | 3D Rotate*
toolbar: *Modify |*

: ***rotate3d***
Prompts you in the command bar:

Select entities to rotate: (Choose one or more entities.)
Entities in set: 1

Select entities to rotate: (Choose more entities, or press Enter to continue.)

Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis/2points] <2points>: (Pick a point, or enter an option.)

Specify the second rotation axis point: (Pick another point.)

Rotation angle or [Base angle/Copy]: (Type a value, or enter an option.)

Command Options

Option	Description
Select entities to rotate	Chooses the entities to be rotated. You can choose one or more entities using any entity selection method. Press Enter to end entity selection.
2 points	Defines the rotation axis by two points; prompts you: Specify the first rotation axis point - pick a point. Specify the second rotation axis point - pick a point
Entity	Uses the geometry of an entity as the axis for rotating the entities; prompts you: Select an arc, circle, ellipse, line, 2D or 3D polyline segment - specify the entity. BricsCAD determines the axis of rotation from the entity: • Arc - the axis of rotation is vertically centered on the arc's center point. • Circle - the axis of rotation is vertically centered on the circle's center point. • Ellipse - the axis of rotation is vertically centered on the ellipses's center point. • Line - the line is the axis of rotation. • 2D polyline linear segment - the segment is the axis of rotation. • 2D polyline arc segment - the axis of rotation is vertically centered on the arc's center point. • 3D polyline segment - the segment is the axis of rotation.
Last	Uses the last entered axis for rotating the entities.

View	Uses the current view as the axis for rotating the entities; prompts you: Specify point on view direction axis - locate the axis of rotation. The axis of rotation is parallel to the direction of view, passing through the picked point.
Xaxis	The x axis is the axis of rotation; prompts you: Specify point on the X axis - locate the axis of rotation. The axis of rotation is parallel to the x axis, passing through the picked point.
Yaxis	The y axis is the axis of rotation; prompts you: Specify point on the Y axis - locate the axis of rotation. The axis of rotation is parallel to the y axis, passing through the picked point.
Zaxis	The z axis is the axis of rotation; prompts you: Specify point on the Z axis - locate the axis of rotation. The axis of rotation is parallel to the z axis, passing through the picked point.
Rotation angle	Specifies the angle by which to rotate the entities; prompts you: Rotation angle or [Base angle/Copy] - Do one of the following: • Type a value. Positive values rotate entities counterclockwise from the positive rotation axis. • Specify two points.
Base angle	Specifies the a new base angle; prompts you: Base angle <0> - Do one of the following: • Type a value. • Specify two points. New angle - Do one of the following: • Type a value. • Specify two points.
Copy	Creates a rotated copy of the selection set.

Procedures

Rotating in 3D

Related Commands

Rotate - rotates entities about a point in 2D planes.
Align - aligns one entity with another.
Snap - rotates the cursor, ortho angle, and grid.
UCS - rotates the coordinate system.

RScript

Reruns the currently loaded SCR script file (short for "repeat script").

Accessing the Command

command bar: *rscript*
transparent: *'rscript*

: ***rscript***

Displays no prompts in the command bar.

This command reruns the currently loaded script file.

This command can be placed at the end of a script file to make it loop repeatedly.

You load SCR script files with the Script command.

You create script files with the RecScript command or through the Notepad text editor.

Command Options

No options.

Related Commands

RecScript - records scripts to SCR files.
StopScript - stops recording the script, and saves it to an SCR file.
Script - loads and runs a SCR script file.
Delay - pauses the execution of the script for the specified number of milliseconds.
Resume - resumes a stopped script.

RtLook

Look around in a 3D scene.

Accessing the Command

command bar: *rtlook*
mouse: Ctrl + Middle Button

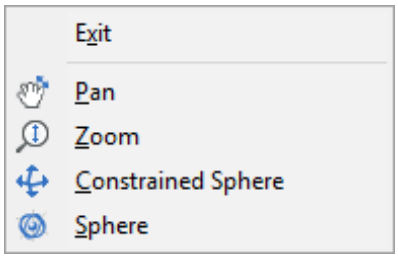
: ***rtlook***
Prompts you in the command bar:

>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to look around in real-time; press Enter to exit command.)
BricsCAD shows the look-around cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtPan

Pans the drawing in real-time.

Accessing the Command

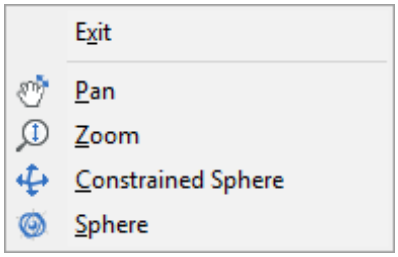
command bar: *rtpan*
transparent: *'rtpan*
menu bar: *View | Real-Time Motion | Real-time Pan*
toolbar: *Real-Time Motion|*

: ***rtpan***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... *(Hold down the mouse's left button, and then move the mouse to pan the drawing in real-time; press Enter to exit command.)*
BricsCAD shows the pan cursor during realtime panning:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

Pan and -Pan - move the drawing within the current viewport.

View and -View - set up named views of drawings.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in realtime.
RtRotCtr - rotates the 3D viewpoint in realtime about a user defined center point.
RtRotF - rotates the 3D viewpoint in realtime.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.

Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRot

Rotates 3D drawings in real-time (short for "real time rotation").

Accessing the Command

command bar: *rtrot*
transparent: *'rtrot*
menu bar: *View | Real-Time Motion | Real-time Constrained Sphere*
toolbar: *Real-Time Motion |*

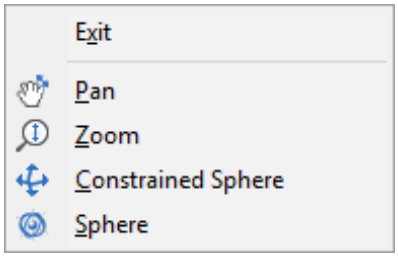
: ***rtrot***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... *(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)*

BricsCAD shows the constrained rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

Navigate - allows to walk or fly through models using the well-known approach used in computer games.
RtLook - look around in a 3D scene.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRotCtr

Rotates 3D drawings in real-time (short for "real time rotation").

Accessing the Command

command bar: *rtrotctr*
transparent: *'rtrotctr*

: ***rtrotctr***
Prompts you in the command bar:

Specify rotation center point:

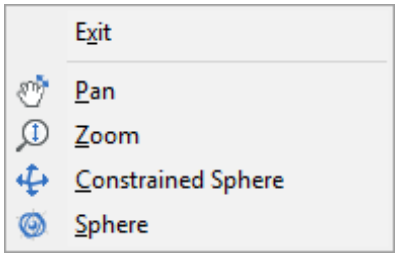
Enter the center point for the rotation: either click a point in the drawing or type the coordinates in the command bar, then press Enter.
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then movethe mouse to rotate the drawing in real-time; press Enter to exit command.)

BricsCAD shows the constrained rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in real time.
RtRotF - rotates the 3D viewpoint in real time.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRotF

Rotates 3D drawings in real-time (short for "real time rotation").

Accessing the Command

command bar: *rtrotf*
transparent: *'rtrotf*
menu bar: *View | Real-Time Motion | Real-time Sphere*
toolbar: *Real-Time Motion|*

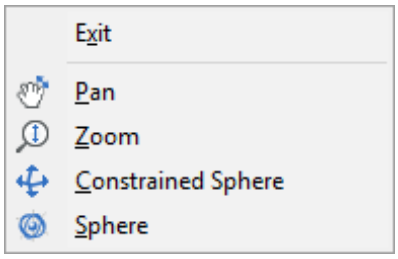
: ***rtrotf***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)

BricsCAD shows the rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRotX

Rotates 3D drawings in real-time about the x axis (short for "real time rotation x-axis").

Accessing the Command

command bar: *rtrotx*
transparent: *'rtrotx*
menu bar: *View | Real-Time Motion | Real-time X*
toolbar: *Real-Time Motion|*

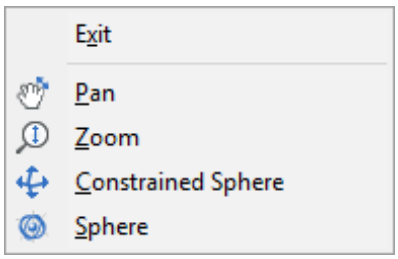
: ***rtrotx***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)

BricsCAD shows the rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in realtime.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRotY

Rotates 3D drawings in real-time about the y axis (short for "real time rotation y-axis").

Accessing the Command

command bar: *rtroty*
transparent: *'rtroty*
menu bar: *View | Real-Time Motion | Real-time Y*
toolbar: *Real-Time Motion|*

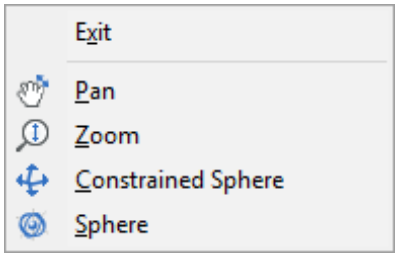
: ***rtroty***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.).

BricsCAD shows the rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in realtime.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtRotZ

Rotates 3D drawings in real-time about the z axis (short for "real time rotation z-axis").

Accessing the Command

command bar: *rtrotz*
transparent: *'rtrotz*
menu bar: *View | Real Time Motion | Real-time Z*
toolbar: *Real-Time Motion |*

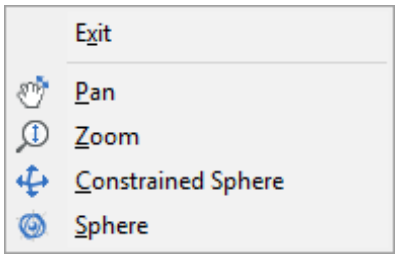
: ***rtrotx***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)

BricsCAD shows the rotation cursor during real-time rotation:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in realtime.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.

Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtUpDown

Move up / down and left / right in a 3D scene.

Accessing the Command

command bar: *RtUpDown*
mouse: Alt + Middle Button

: ***rtupdown***
Prompts you in the command bar:

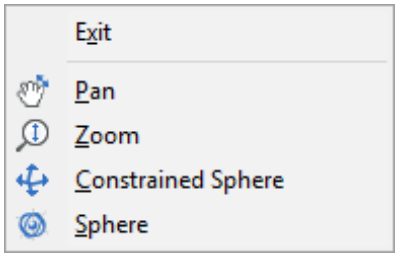
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to move up / down and left / right in real-time; press Enter to exit command.)

BricsCAD shows the up-down cursor during real-time movement:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

RtLook - Look around in a 3D scene.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtWalk - walk left / right and forward / backward in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtWalk

Walk left / right and forward / backward in a 3D scene.

Accessing the Command

command bar: *rtwalk*
mouse: Alt + Left Button

: ***rtwalk***
Prompts you in the command bar:

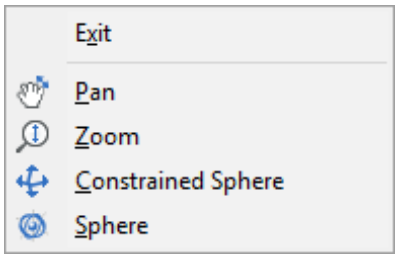
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to walk left / right and forward / backward in real-time; press Enter to exit command.)

BricsCAD shows the walk cursor during real-time movement:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

Navigate - allows to walk or fly through models using the well-known approach used in computer games.
RtLook - look around in a 3D scene.
RrRotF - rotates the 3D viewpoint in real time.
RtRotCtr - rotates the 3D viewpoint in real time about a user defined center point.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.

Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.
Zoom - increases and decreases the size of the drawing within the current viewport.
RtZoom - performs real-time zooming of the drawing.

RtZoom

Zooms the drawing in real-time.

Accessing the Command

command bar: *rtzoom*
transparent: *'rtzoom*
menu bar: *View | Real-Time Motion | Real-time Zoom*
toolbar: *Real-Time Motion |*

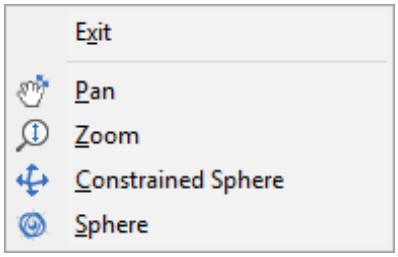
: ***rtzoom***
Prompts you in the command bar:
>>Press ENTER or Esc to complete, or right-click to display context menu... (Hold down the mouse's left button, and then move the mouse to zoom the drawing in real-time; press Enter to exit command.)

BricsCAD shows the zoom cursor during realtime zooming:

Command Options

Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options



Option	Description
Exit	Exits the command.
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 RtRotctr the command.
Sphere	Rotates the 3D drawing in real-time Starts the RtRot command.

Related Commands

Zoom - increases and decreases the size of the drawing within the current viewport.
Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.

View and -View - set up named views of drawings.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
RtLook - Look around in a 3D scene.
RtRot - rotates the 3D viewpoint in realtime.
RtRotCtr - rotates the 3D viewpoint in realtime about a user defined center point.
RtRotF - rotates the 3D viewpoint in realtime.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.
RtUpDown - move up / down and left / right in a 3D scene.
RtWalk - walk left / right and forward / backward in a 3D scene.

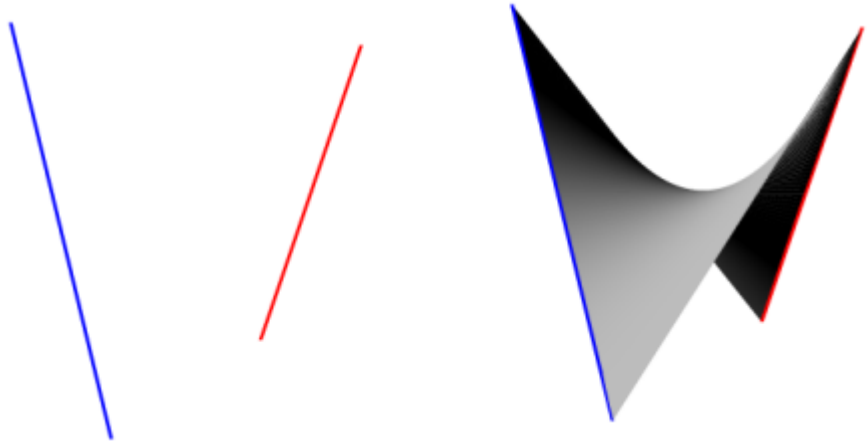
RuleSurf

Creates 3D mesh surfaces by connecting two separate linear entities.

Accessing the Command

command bar: *rulesurf*
menu bar: *Model | Meshes | Ruled Surface*
toolbar: *Meshes | 3D Surfaces |*

: ***rulesurf***
Prompts you in the command bar:
Select entity to define first end of ruled surface: (Select an open object.)
Select entity to define other end: (Select another open object.)



Ruled surface created between two lines.

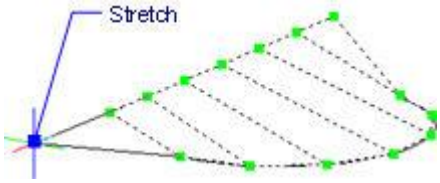
Command Options

Option	Description
Select entity to define first end of ruled surface	Specifies the first boundary entity; choose any open object, such as line, arc, polyline, or spline. Closed entities and 3D entities are not accepted; prompts you: Unable to use the selected entity. Please pick again.
Select entity to define other end	Specifies the second boundary entity.

Grips Editing

Ruled surfaces can be edited directly through grips:

1. Select the rulesurf. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
TabSurf - draws tabulated surfaces as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

RunAsLevel

Allows to run BricsCAD in a lower license level.

Accessing the Command

command bar: *runaslevel*

: ***runaslevel***
Prompts you in the command bar:

Prompt: (User response)
Instructions

Specify product [BricsCAD/Communicator/SheetMetal/BIM] <BricsCAD>: (Choose an option.)

Command Options

Option	Description
BricsCAD	Prompts you: Run BricsCAD as [Classic/Pro/PLatinum] <Platinum> - choose an option. • Classic: run BricsCAD in Classic mode with a Pro or Platinum license. • Pro: run Bricscad in Pro mode with a Platinum license; or restore the Pro license level. • Platinum: restore the Platinum license level. The new level will be in use after restarting BricsCAD. The selected level is saved through the *RunAsLevel* user preference.
Communicator	Prompts you: Run Communicator with license [No licence/Trial/Full] <Trial> - choose an option. • No license: Communicator is not available. • Trial: Communicator runs in trial mode. • Full: restores the Communicator license. The new level will be in use after restarting BricsCAD. The selected level is saved through the *UseCommunicator* user preference.
SheetMetal	Prompts you: Run SheetMetal with license [No licence/Trial/Full] <Full> - choose an option. • No license: Sheet Metal commands are not available. • Trial: Shee tMetal runs in trial mode. • Full: restores the Sheet Metal license. The new level will be in use after restarting BricsCAD. The selected level is saved through the *UseSheetMetal* user preference.
BIM	Run BIM with license [No licence/Trial/Full] <Full> - choose an option. • No license: BIM commands are not available. • Trial: BIM runs in trial mode. • Full: restores the BIMl license. The new level will be in use after restarting BricsCAD. The selected level is saved through the *UseBIM* user preference.

Related Commands

LicenseManager - displays the License Manager dialog, summarizing your available licenses (BricsCAD / BIM / Sheet Metal / Communicator).

S

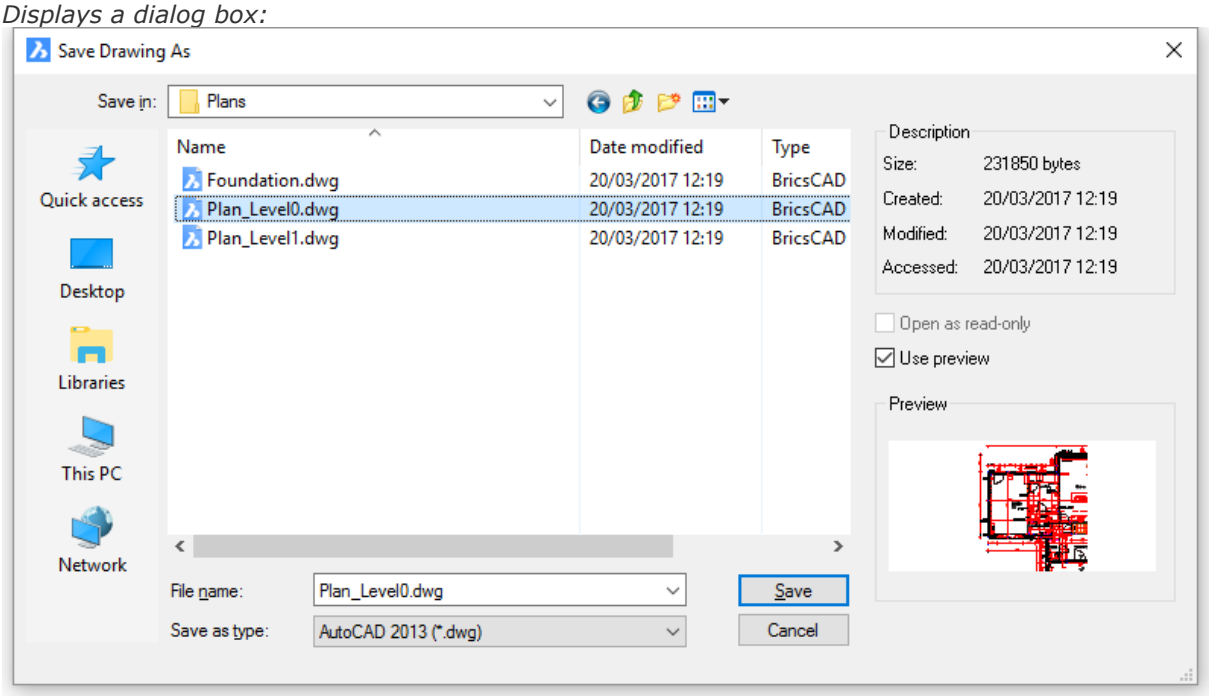
Save

Saves drawings in DWG drawing, DWT template, and DXF interchange formats; always displays a dialog box.

Accessing the Command

command bar: *save*
shortcut: *Ctrl+S*
aliases: *sa*

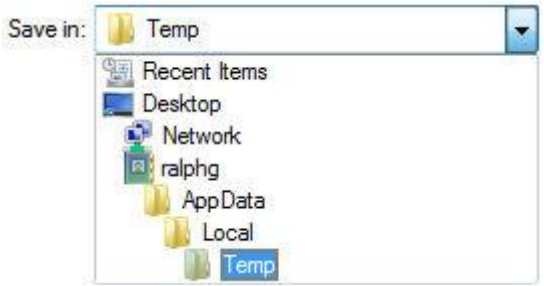
: ***save***
Displays a dialog box:



Enter a file name, and then click Save.

Use the **QSave** command to save drawings without seeing this dialog box.

Command Options

Option	Description
	Selects the drive and folder in which to save the drawing. Click the droplist to access additional locations.

File name: Drawing1.dwg C:\Users\ralphg\AppData\Local\Temp\Drawing C:\Program Files\Bricsys\Bricscad\yard.dwg \\Ralph\d\CAD\dwg\4015\4015a.dwg	Specifies the name by which to save the drawing: • Accept the default *drawing.dwg* file name. • Enter another name. • Click the droplist to access names of previously-save drawings.
AutoCAD 2013 (*.dwg) AutoCAD 2013 ASCII DXF (*.dxf) AutoCAD 2013 Binary DXF (*.dxf) AutoCAD 2010 (*.dwg) AutoCAD 2010 ASCII DXF (*.dxf) AutoCAD 2010 Binary DXF (*.dxf) AutoCAD 2007 (*.dwg) AutoCAD 2007 ASCII DXF (*.dxf) AutoCAD 2007 Binary DXF (*.dxf) AutoCAD 2004 (*.dwg) AutoCAD 2004 ASCII DXF (*.dxf) AutoCAD 2004 Binary DXF (*.dxf) AutoCAD 2000 (*.dwg) AutoCAD 2000 ASCII DXF (*.dxf) AutoCAD 2000 Binary DXF (*.dxf) AutoCAD Release 14 (*.dwg) AutoCAD Release 14 ASCII DXF (*.dxf) AutoCAD Release 14 Binary DXF (*.dxf) AutoCAD Release 13 Drawing (*.dwg) AutoCAD Release 13 ASCII DXF (*.dxf) AutoCAD Release 13 Binary DXF (*.dxf) AutoCAD Release 11/12 Drawing (*.dwg) AutoCAD Release 11/12 ASCII DXF (*.dxf) AutoCAD Release 11/12 Binary DXF (*.dxf) AutoCAD Release 10 ASCII DXF (*.dxf) AutoCAD Release 10 Binary DXF (*.dxf) AutoCAD Release 9 ASCII DXF (*.dxf) Drawing Template (*.dwt)	Specifies the format of the saved file. Accept the default (AutoCAD 2007 .dwg) or click the droplist to choose another format. There are three basic formats: • DWG - drawing file. • DXF - interchange file (ASCII or Binary). • DWG - template file.
☑ Use Preview	Includes a preview image with the drawing file. The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing.
Cancel	Cancels the command; does not save the drawing.

Procedures

Working with document tabs

Related Commands

Export - exports drawings in a variety of vector and raster formats.
QSave - saves the drawing without displaying the Save dialog box.
SaveAll - saves all open drawings.
SaveAs - saves drawings by another name.
SaveAsR12 - saves drawing in older formats.
SaveFileFolder - opens the SAVEFILEPATH location in the system file browser.
SecurityOptions - sets a password to protect a drawing

SaveAll

Saves all open drawings.

Accessing the Command

command bar: *saveall*

: ***saveall***

Displays no prompt in the command bar.

Caution: this command saves all drawings with no 'Are you sure?' check. BricsCAD may save drawings that you did not intend to be saved, and so could overwrite files.

Command Options

No options.

Procedures

Working with document tabs

Related Commands

Save - saves the current drawing through a dialog box.
QSave - saves the drawing without displaying the Save dialog box.
SaveAs - saves drawings by another name.
SaveAsR12 - saves drawing in older formats.
Export - exports drawings in a variety of vector and raster formats.

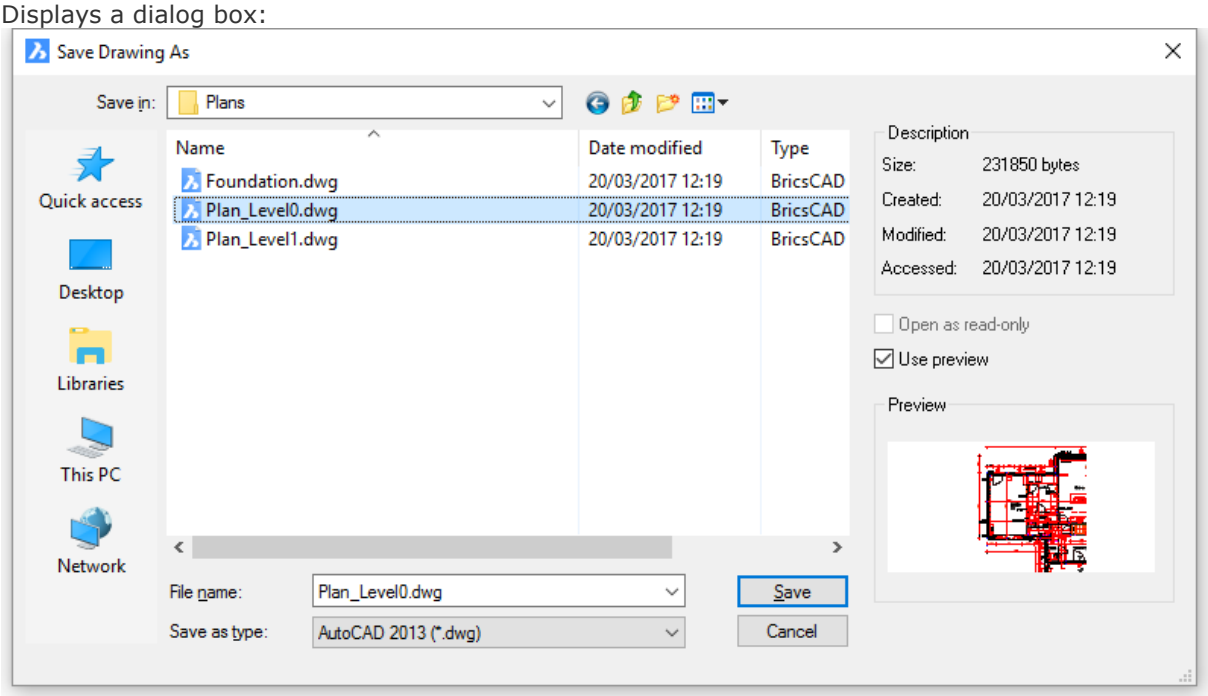
SaveAs

Saves drawings in DWG drawing, DWT template, and DXF interchange formats; always displays a dialog box.

Accessing the Command

command bar: *saveas*

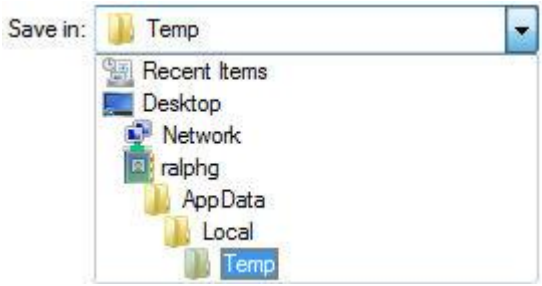
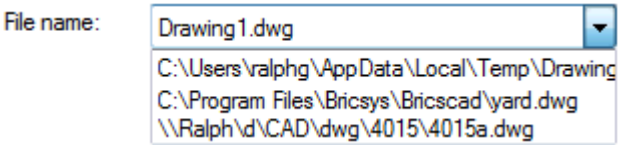
: ***saveas***
Displays a dialog box:



Enter a file name, and then click Save.

Use the **QSave** command to save drawings without seeing this dialog box.

Command Options

Option	Description
	Selects the drive and folder in which to save the drawing. Click the droplist to access additional locations.
	Specifies the name by which to save the drawing: • Accept the default drawing.dwg file name. • Enter another name.

	• Click the droplist to access names of previously-save drawings.
AutoCAD 2013 (*.dwg) AutoCAD 2013 ASCII DXF (*.dxf) AutoCAD 2013 Binary DXF (*.dxf) AutoCAD 2010 (*.dwg) AutoCAD 2010 ASCII DXF (*.dxf) AutoCAD 2010 Binary DXF (*.dxf) AutoCAD 2007 (*.dwg) AutoCAD 2007 ASCII DXF (*.dxf) AutoCAD 2007 Binary DXF (*.dxf) AutoCAD 2004 (*.dwg) AutoCAD 2004 ASCII DXF (*.dxf) AutoCAD 2004 Binary DXF (*.dxf) AutoCAD 2000 (*.dwg) AutoCAD 2000 ASCII DXF (*.dxf) AutoCAD 2000 Binary DXF (*.dxf) AutoCAD Release 14 (*.dwg) AutoCAD Release 14 ASCII DXF (*.dxf) AutoCAD Release 14 Binary DXF (*.dxf) AutoCAD Release 13 Drawing (*.dwg) AutoCAD Release 13 ASCII DXF (*.dxf) AutoCAD Release 13 Binary DXF (*.dxf) AutoCAD Release 11/12 Drawing (*.dwg) AutoCAD Release 11/12 ASCII DXF (*.dxf) AutoCAD Release 11/12 Binary DXF (*.dxf) AutoCAD Release 10 ASCII DXF (*.dxf) AutoCAD Release 10 Binary DXF (*.dxf) AutoCAD Release 9 ASCII DXF (*.dxf) Drawing Template (*.dwt)	Specifies the format of the saved file. Accept the default (AutoCAD 2007 .dwg) or click the droplist to choose another format. There are three basic formats: • DWG - drawing file. • DXF - interchange file (ASCII and Binary). • DWT - template file.
☑ Use Preview	Includes a preview image with the drawing file. The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing.
Cancel	Cancels the command; does not save the drawing.

Procedures

Working with document tabs

Related Commands

Export - exports drawings in a variety of vector and raster formats.
QSave - saves the drawing without displaying the Save dialog box.
Save - saves the current drawing through a dialog box.
SaveAll - saves all open drawings.
SaveAsR12 - saves drawing in older formats.
SecurityOptions - sets a password to protect a drawing

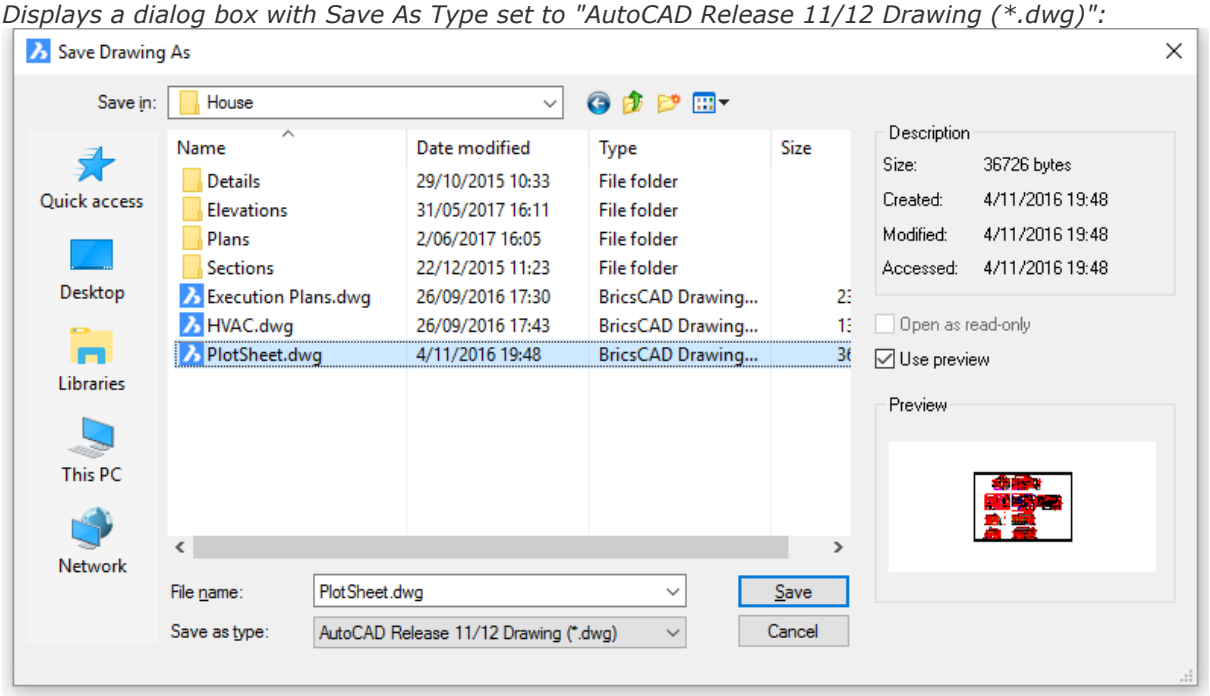
SaveAsR12

Saves drawings in DWG R12 format through a dialog box.

Accessing the Command

command bar: *saveas12*

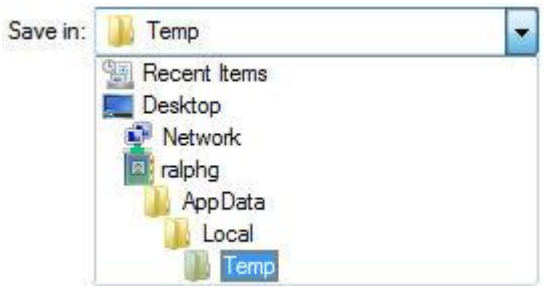
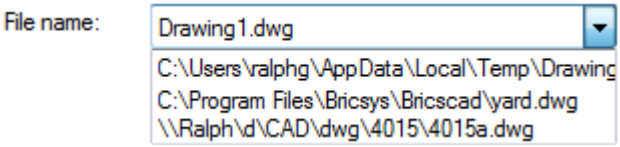
: ***saveasr12***
Displays a dialog box with Save As Type set to "AutoCAD Release 11/12 Drawing (.dwg)":*



Enter a file name, and then click Save.

To save the drawing in other formats, click the Save As Type droplist.

Command Options

Option	Description
	Selects the drive and folder in which to save the drawing. Click the droplist to access additional locations.
	Specifies the name by which to save the drawing: • Accept the default *drawingn.dwg* file name. • Enter another name.

	• Click the droplist to access names of previously-save drawings.
AutoCAD 2013 (*.dwg) AutoCAD 2013 ASCII DXF (*.dxf) AutoCAD 2013 Binary DXF (*.dxf) AutoCAD 2010 (*.dwg) AutoCAD 2010 ASCII DXF (*.dxf) AutoCAD 2010 Binary DXF (*.dxf) AutoCAD 2007 (*.dwg) AutoCAD 2007 ASCII DXF (*.dxf) AutoCAD 2007 Binary DXF (*.dxf) AutoCAD 2004 (*.dwg) AutoCAD 2004 ASCII DXF (*.dxf) AutoCAD 2004 Binary DXF (*.dxf) AutoCAD 2000 (*.dwg) AutoCAD 2000 ASCII DXF (*.dxf) AutoCAD 2000 Binary DXF (*.dxf) AutoCAD Release 14 (*.dwg) AutoCAD Release 14 ASCII DXF (*.dxf) AutoCAD Release 14 Binary DXF (*.dxf) AutoCAD Release 13 Drawing (*.dwg) AutoCAD Release 13 ASCII DXF (*.dxf) AutoCAD Release 13 Binary DXF (*.dxf) AutoCAD Release 11/12 Drawing (*.dwg) AutoCAD Release 11/12 ASCII DXF (*.dxf) AutoCAD Release 11/12 Binary DXF (*.dxf) AutoCAD Release 10 ASCII DXF (*.dxf) AutoCAD Release 10 Binary DXF (*.dxf) AutoCAD Release 9 ASCII DXF (*.dxf) Drawing Template (*.dwt)	Specifies the format of the saved file. Accept the default (AutoCAD 2007 .dwg) or click the droplist to choose another format. There are three basic formats: • DWG - drawing file. • DXF - interchange file (ASCII and Binary). • DWG - template file.
☑ Use Preview	Includes a preview image with the drawing file. The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing.
Cancel	Cancels the command; does not save the drawing.

Related Commands

Save - saves the current drawing through a dialog box.
QSave - saves the drawing without displaying the Save dialog box.
SaveAll - saves all open drawings.
SaveAs - saves drawings by another name.
Export - exports drawings in a variety of vector and raster formats.

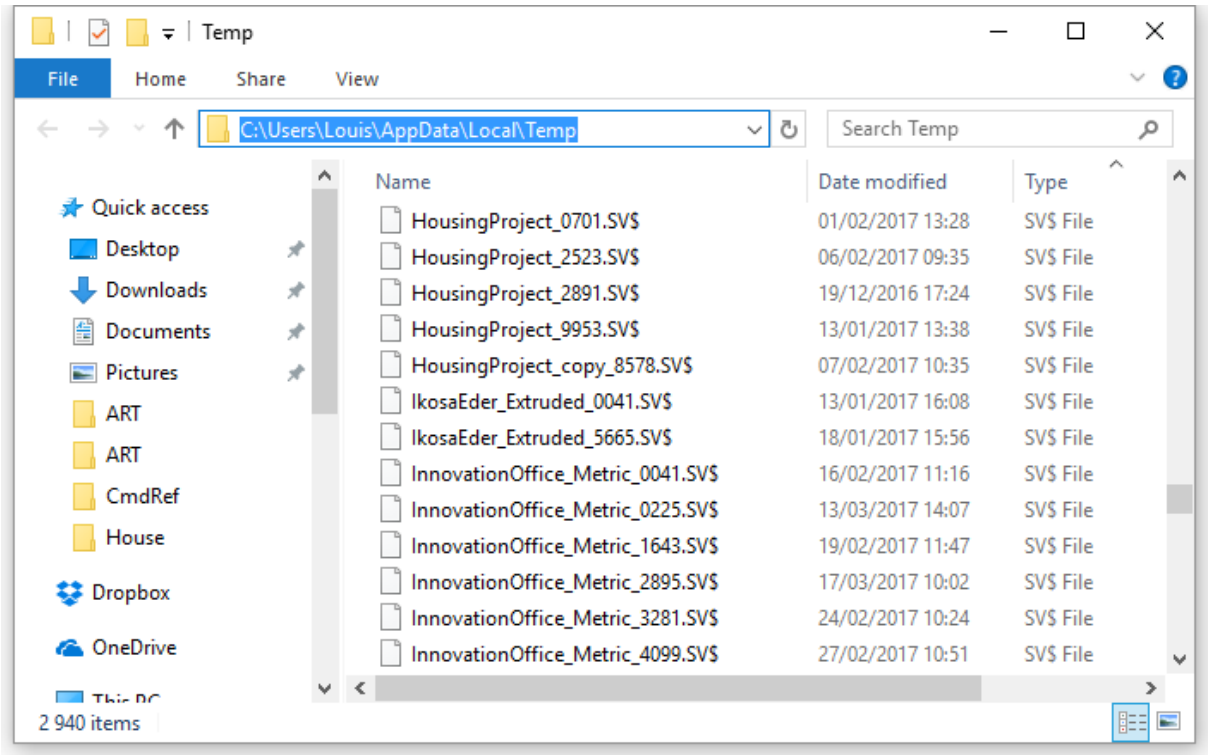
SaveFileFolder

Opens the SAVEFILEPATH location in the system file browser.

Accessing the Command

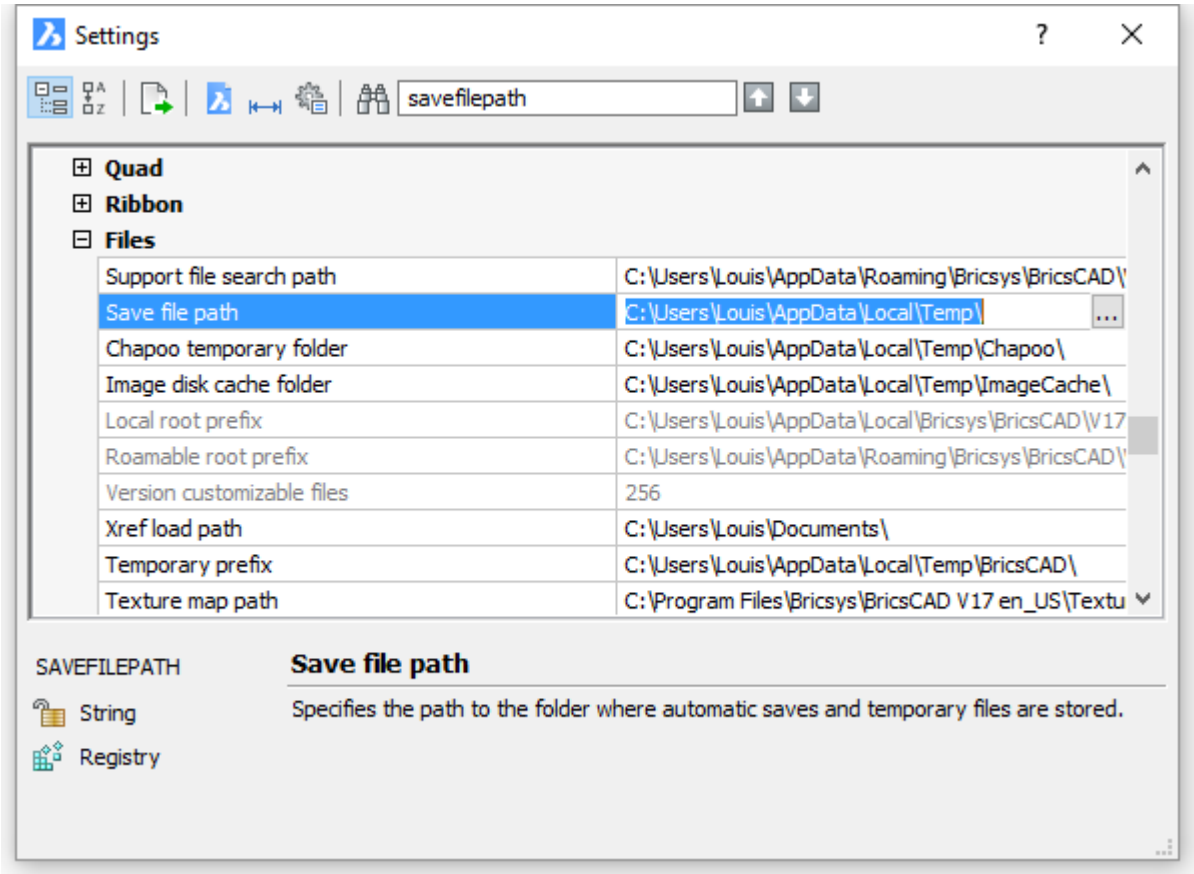
command bar: *savefilefolder*

: ***savefilefolder***
Opens a File dialog box:



A *.SV$ backup file of all open drawings is saved in the SAVEFILEPATH location a the interval defined by the SAVETIME system variable.

The location is defined by the SAVEFILEPATH setting:



Click the *Browse* button of the *Save file path* system variable to select a different folder.

Command Options

No command options.

Related Commands

Save - saves drawings in DWG drawing, DWT template, and DXF interchange formats.

Scale

Resizes entities.

Accessing the Command

command bar: *scale*
alias: *sc*
menu bar: *Modify | Scale*
toolbar: *Modify |*

: ***scale***
Prompts you in the command bar:

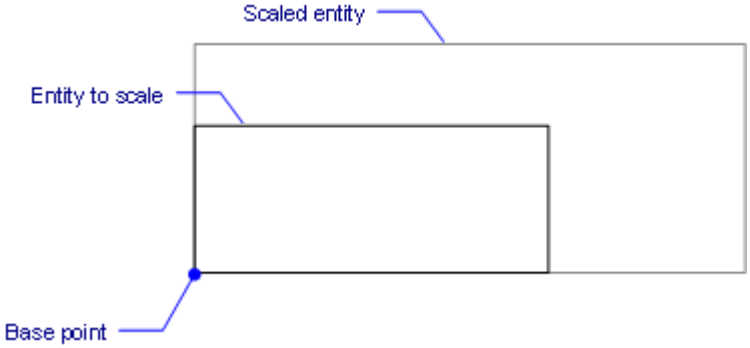
Select entities to scale: (Choose one or more entities.)

Entities in set: 1

Select entities to scale: (Choose more entities, or press Enter to continue.)

Base point: (Pick a point or enter x,y,z coordinates.)

Copy/Base scale/<Scale factor>: (Enter a number, or type C or B.)



Command Options

Option	Description
Select entities to scale	Chooses the entities to be resized. Use any selection method.
Base point	Specifies the base point. This is the point from which the entities "grow" or "shrink." Pick a point in the drawing, or enter x,y,z coordinates. Use entity snap to pick a geometric feature accurately.
Scale factor	Specifies the scale factor: • Enter a number smaller than 1 to shrink the entities. • Enter 1 to leave the entities the same size. • Enter a number larger than 1 to make the entities larger.
Copy	Creates a scaled copy of the selection set.

Base scale	Resizes entities using a base or reference scale; prompts you: Base scale - enter a scale factor or pick two points. New scale - enter the new scale factor. Use this option to scale entities relative to other entities. For example, pick two points on an existing entity. When you pick two points, BricsCAD prompts you: Second point - pick another point.

Procedures

To scale a selection set
Scaling an entity using a base scale
To scale entities using grips

Related Commands

Align - scales one entity with another.
Insert - scales blocks.
Print - scales drawings upon plotting.

ScaleListEdit

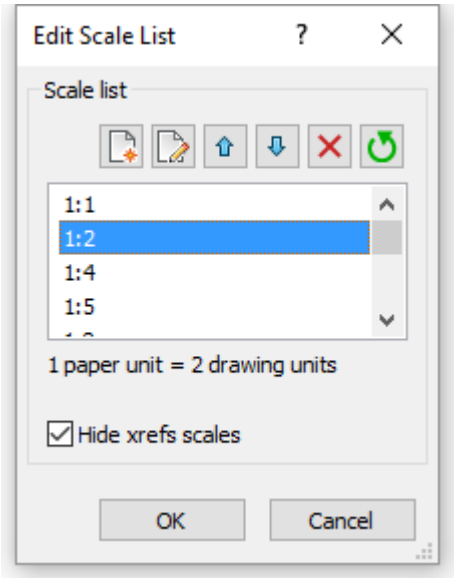
Edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport.
Defines the available annotation scales for annotative entities.

Accessing the Command

command bar: *scalelistedit*

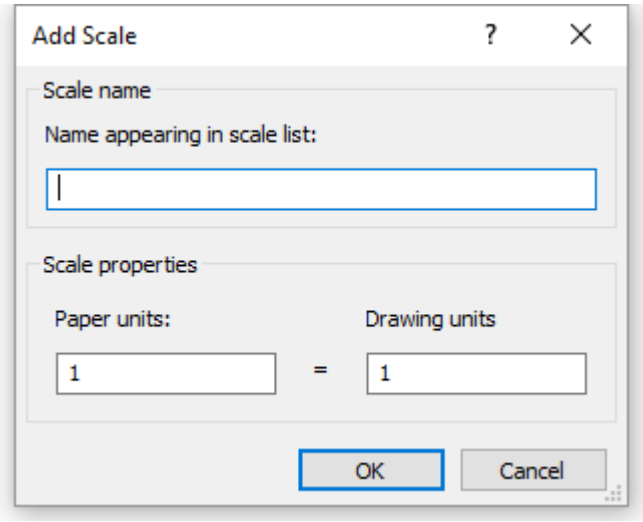
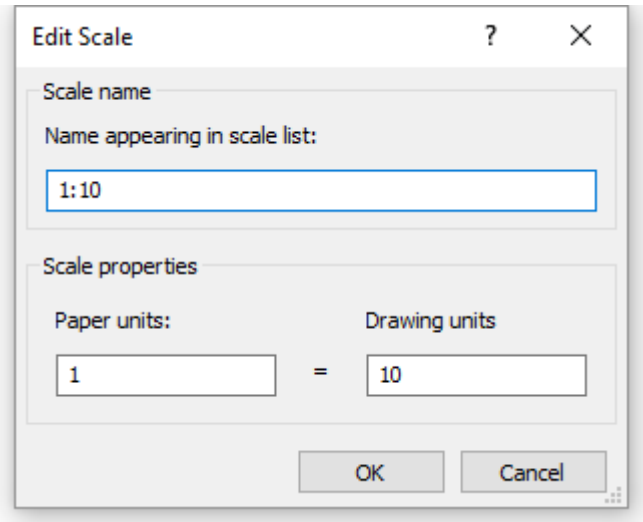
: ***scalelistedit***

Displays a dialog box:



Command Options

Option	Description
Hide xrefs scales	Sets the HIDEXREFSCALES system variable. If unchecked, scales in the attached xrefs that are not available in the current drawing are listed here and in the list of available scales on the Page Setup and Print dialogs and in the *Standard Scale* property of a layout viewport.
Add Scale	Displays a dialog box:

	 Name appearing in scale list: type a name in the *Scale name* field. Scale properties: specify the scale factor Press the *OK* button to add the scale to the list.
Edit Scale	Displays a dialog box:  Name appearing in scale list: type a name in the *Scale name* field. Scale properties: specify the scale factor Press the *OK* button to save the changes.
Move Up	Move the selected scale up in the scale list.
Move Down	Move the selected scale down in the scale list.
Delete	Delete the selected scale.
Reset	Reloads hard-coded default scales. Deletes all unreferenced user defined scales.
OK	Saves the changes in the drawing and closes the dialog box.
Cancel	Closes the dialog box. The changes are not saved.

Related Commands

ObjectScale - adds or removes supported scales for annotative entities through a dialog box.
-ObjectScale - adds or removes supported scales for annotative entities at the command line.
Print - prints drawings to plotters and files through a dialog box.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.
-ScaleListEdit - edits the scale list of a drawing in the command bar.

-ScaleListEdit

Edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport.
Defines the available annotation scales for annotative entities.

Accessing the Command

command bar: *-scalelistedit*

: ***-scalelistedit***
Prompts you in the command bar:

Enter option [?/Add/Delete/Reset/Exit] <Add>: (Enter an option or press Enter to add a new scale)
Instructions

Next Prompt: (User response)

Command Options

Option	Description
?	Displays the current scale list in the command bar. Press the F2 function key to open the Prompt History window.
Add	Creates a new scale Prompts you in the command bar: Enter new scale name - type a name for the scale in the command bar. The scale name appears in: • the *Edit Scale List* dialog box (see the ScaleListEdit command) • the Annotation Scale list in the *Properties Bar*. Enter scale ratio - type the scale ration in the command bar. The following formats are accepted, e.g. to define a scale of 1 paper unit = 10 drawing units. • 1:10 • 1=10
Delete	Removes a scale from the scale list. Referenced scales cannot be deleted. Prompts you in the command bar: Enter scale name to delete or * to delete all unused scales - Do one of the following • Type the name of the scale to delete. • type * to delete all unreferenced scales.
Reset	Reloads hard-coded default scales. Deletes all unreferenced user defined scales.
Exit	Concludes the command.

Related Commands

Print - prints drawings to plotters and files through a dialog box.
PageSetup - creates and edits page setups for plotting drawings in the Drawing Explorer.
ScaleListEdit - edits the scale list of a drawing in a dialog box.

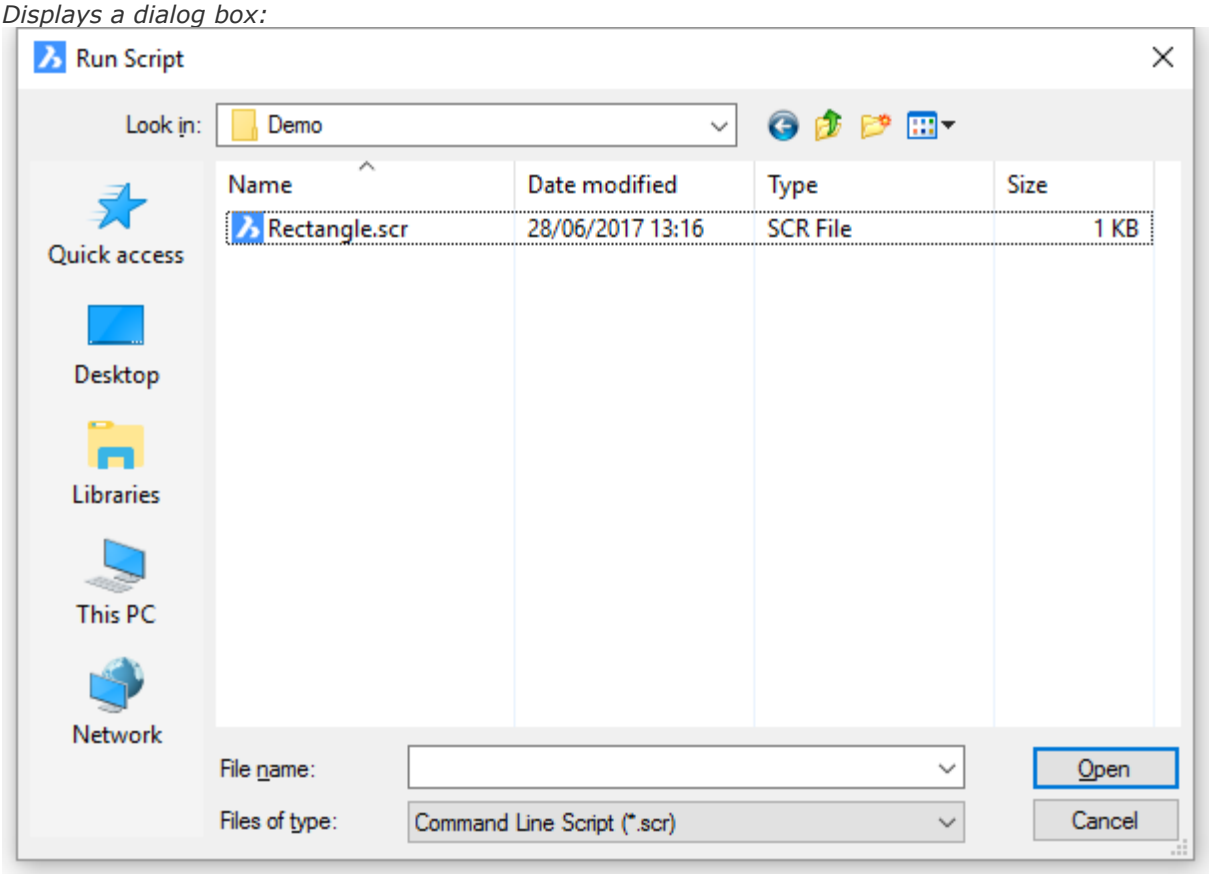
Script

Loads and runs SCR script files through a dialog box.

Accessing the Command

command bar: *script*
transparent: *'script*
alias: *scr*
menu bar: *Tolls | Run Script*
toolbar: *Tools |*

: ***script***
Displays a dialog box:



Select an SCR file, and then click Open. *The script begins running immediately.*

To rerun the script, use the RScript command. For more information on writing scripts, see Customizing BricsCAD.

Command Options

Option	Description
Look in	Chooses the drive and folder name.
File name	Specifies the name of the SCR file.
Open	Opens the SCR file, and then runs the script.

Related Commands

RecScript - records scripts to SCR files.
StopScript - stops recording the script, and saves it to an SCR file.
RScript - reruns the script.
Delay - pauses the execution of the script for the specified number of milliseconds.
Resume - resumes a stopped script.

Scrollbar

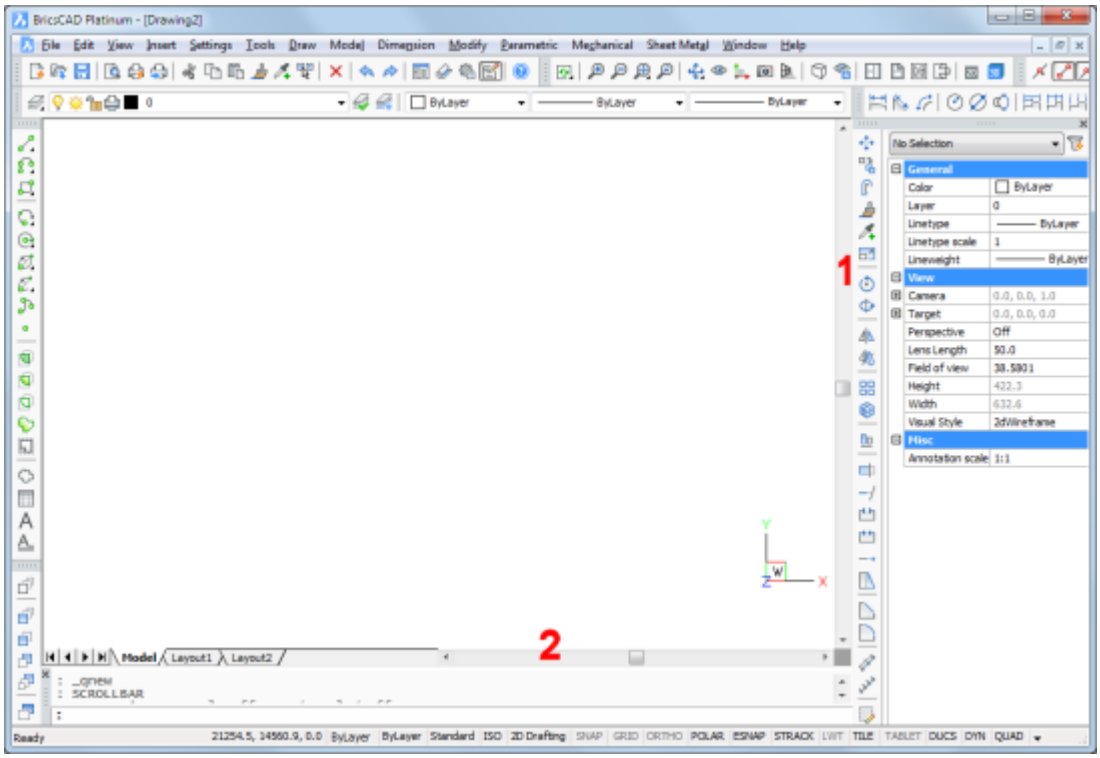
Toggles the display of the scroll bars.

Accessing the Command

command bar: *scrollbar*
transparent: *'scrollbar*
menu bar: *View | Scrollbars*

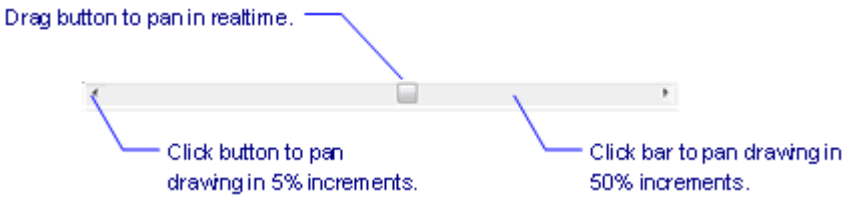
: ***scrollbar***
Prompts you in the command bar:

WNDLSCRL is currently on: OFF/Toggle/<On>: ***(Enter an option.)***



1. Vertical scrollbar
2. Horizontal scrollbar

Scroll bars are useful for panning the drawing without using the **Pan** command:



The only reason to turn off scroll bars is to gain a bit more drawing area.

Command Options

Option	Description
OFF	Turns off the scroll bars.
Toggle	Toggles the display of scroll bars: • If on, turns them off. • If off, turns them on.
On	Turns on the scroll bars.

Related Commands

CommandLine - shows the command line.
CommandLineHide - hides the command line.
GraphScr - brings the graphics window to the foreground.
StatBar - toggles the display of the status bar.
TextScr - brings the text window to the foreground.
Toolbar and -Toolbar - toggle the display of toolbars.

Section

Creates section planes of 3D solids; the results are region entities.

Accessing the Command

command bar: *section*
alias: *sec*
menu bar: *Model | 3D solids| Section*
toolbar: *3D Solids |*

: ***section***

Prompts you in the command bar:

Select entities: (Choose one or more 3D solids.)

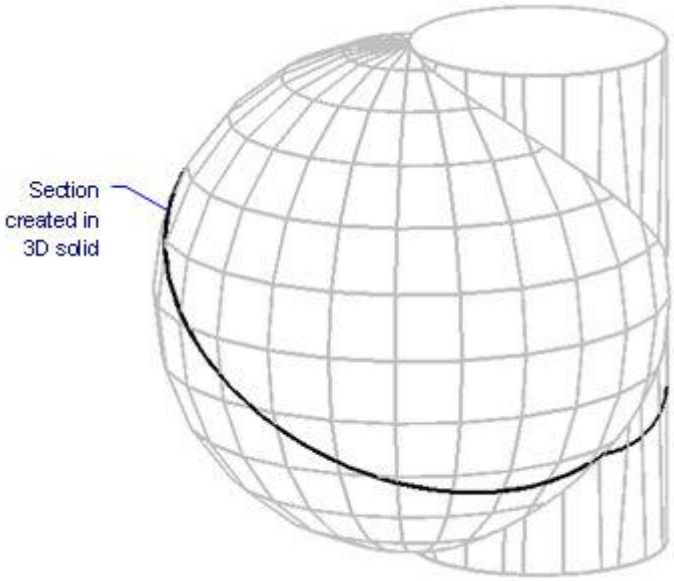
Entities in set: 1

Select entities: (Choose more 3D solids, or press Enter to continue.)

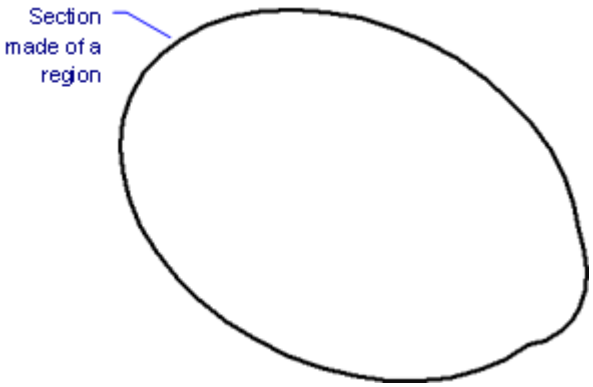
Specify first point on section plane or Object/Zaxis/View/XY/YZ/ZX: (Enter an option.)

Specify a point on the current view plane <0,0,0>: (Pick a point, or enter x,y,z coordinates.)

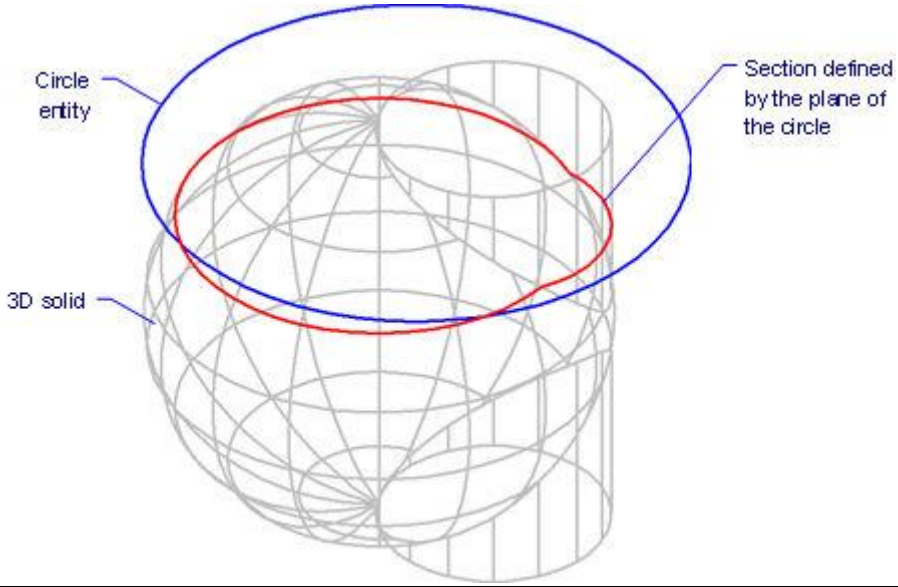
Section made in the current view plane:

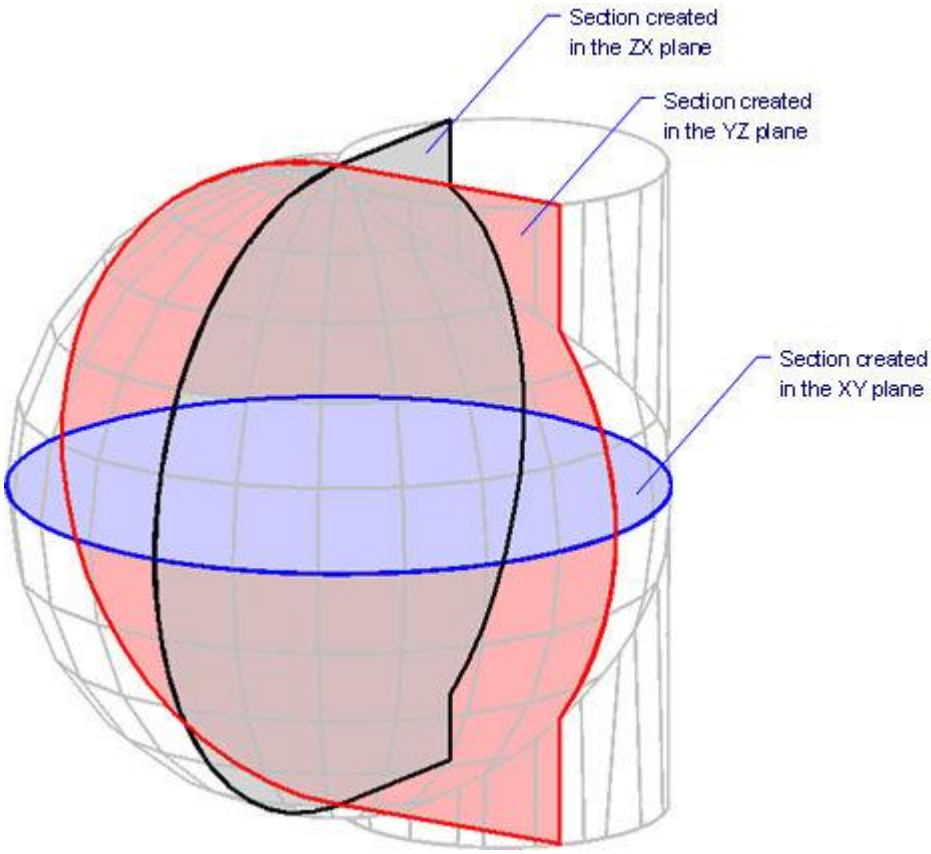


Sections are made of regions, and can be manipulated as independent entities:



Command Options

Option	Description
Select entities	Specifies the entities from which the section is created. Choose one or more 3D solids; this command does not work with 2D regions.
Specify first point on section plane	Pick a point to specify the plane in which the section lies; prompts you: Specify second point on plane - pick another point. Specify third point on plane - pick a third point (three points define a plane).
Object	Creates the section from entities that intersect the 3D solid; prompts you: Select a circle, ellipse, arc, 2D-spline, or 2D-polyline - choose an entity. You can also use elliptical arcs. The entity defines the plane through which to create the section. 
Zaxis	Creates a section in the plane defined by one point on the plane, and a second on the plane's normal (z axis); prompts you: Specify a point on the section plane - pick a point to define the location of the section plane. Specify a point on the Z-axis (normal) of the plane - pick another point to locate the normal.
View	Creates a section in the plane of the current 3D viewpoint; prompts you: Specify a point on the current view plane - pick a point to define the height of the section. This is probably the easiest of the options.
XY	Creates a section parallel to the x,y plane; prompts you: Specify a point on the XY-plane - pick a point to define the height of the section above or below the x,y plane.
YZ	Creates a section parallel to the y,z plane; prompts you: Specify a point on the YZ-plane - pick a point to define the height of the section above or below the y,z plane.
ZX	Creates a section parallel to the z,x plane; prompts you: Specify a point on the ZX-plane - pick a point to define the height of the section above or below the z,x plane.



Related Commands

Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Slice - cuts 3D solids into two parts.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.

ClipDisplay - toggles the Clip Display property of a section plane or a BIM section entity.
BimSection - creates a BIM section entity.
BimSectionOpen - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.
LiveSection - switches the *Live Section* property of a section plane on or off.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.
SectionPlaneToBlock - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

SectionPlane

Creates a section entity that creates sections of 3D solids.

Accessing the Command

command bar: *sectionplane*
menu bar: *Model* | *Sections* | *Section Plane*
toolbar: *Sections* |

: ***sectionplane***
Prompts you in the command bar:

Specify start point or [select Face/Draw/Orthographic]: (specify the start point of the section plane)

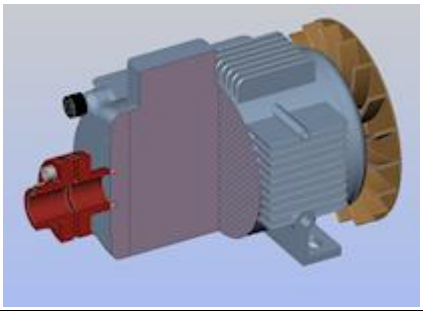
Specify through point: (specify a second point)

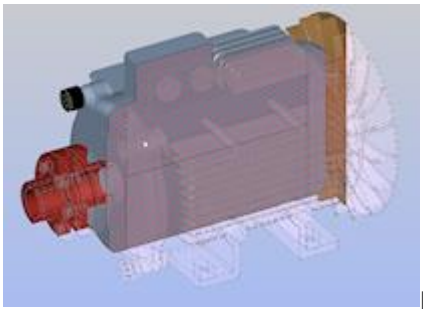
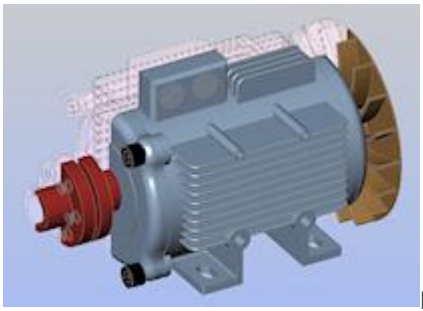
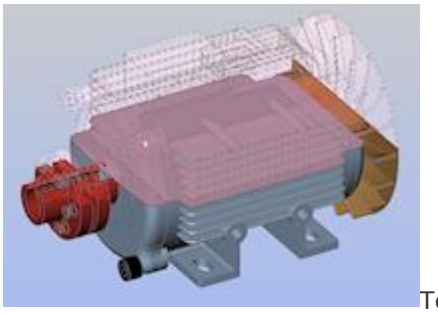
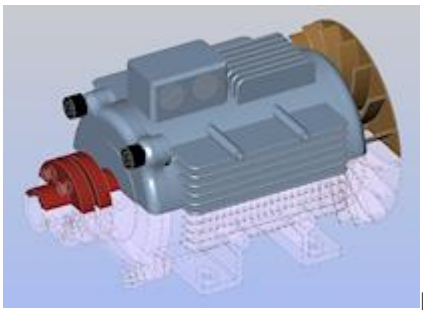
The section plane is defined through the two points and perpendicular to the XY-plane of the current UCS.

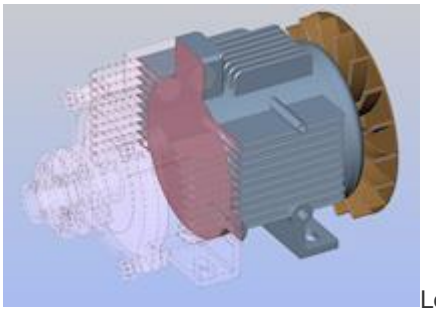
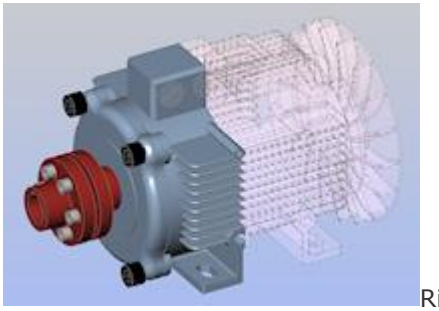
The part of the 3D solids at the right hand side of the section plane is cut away.

Newly created section planes have the *Clip Display* status *ON*.

Command Options

Option	Description
Select Face	Select the planar face of a 3D solid. The section plane coincides with the selected face. The *Live Section* property is turned on automatically.
Draw	Define 2 points to define a planar section plane; define multiple points to create boundary section plane for jogged sections. You are prompted in the command bar: Specify start point: Specify the first point. Specify next point: Specify the second point. Specify next point or ENTER to finish: Specify the next point(s) of the section plane. Press Enter to stop specifying points. Specify the direction of the section entity: Specify a point to define the view direction of the section plane. The section plane is created through the specified points and perpendicular to the current UCS. The Image below shows the result of a 4 point boundary section plane. 

Orthographic	A section plane entity with an orthographic orientation relative to the current UCS is created. The options are: *Front*, *Back*, *Top*, *Bottom*, *Left* and *Right*. Orthographic section planes are created through the center of the bounding box that contains all 3D entities in the drawing. The *Live Section* property is turned on automatically. The images below show the result of the various orthographic section planes, including the cut-away geometry.
	 Front
	 Back
	 Top
	 Bottom

	 Left
	 Right

Operations

Align a section plane to a face
To create a vertical section plane
To create a horizontal section plane
To create a vertical jogged section plane
To create a horizontal jogged section plane
To create a orthographic section plane
To edit the properties of a section plane
To modify the size and position of a section plane

Related Commands

ClipDisplay - toggles the *Clip Display* property of a section plane or a BIM section entity.
LiveSection - switches the *Live Section* property of a section plane on or off.
Section - creates section planes of 3D solids; the results are region entities.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.
SectionPlaneToBlock - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

BimSection - creates a BIM section entity.
BimSectionOpen - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.
BimSectionUpdate - exports or updates the result of a BIM section in a drawing.

SectionPlaneSettings

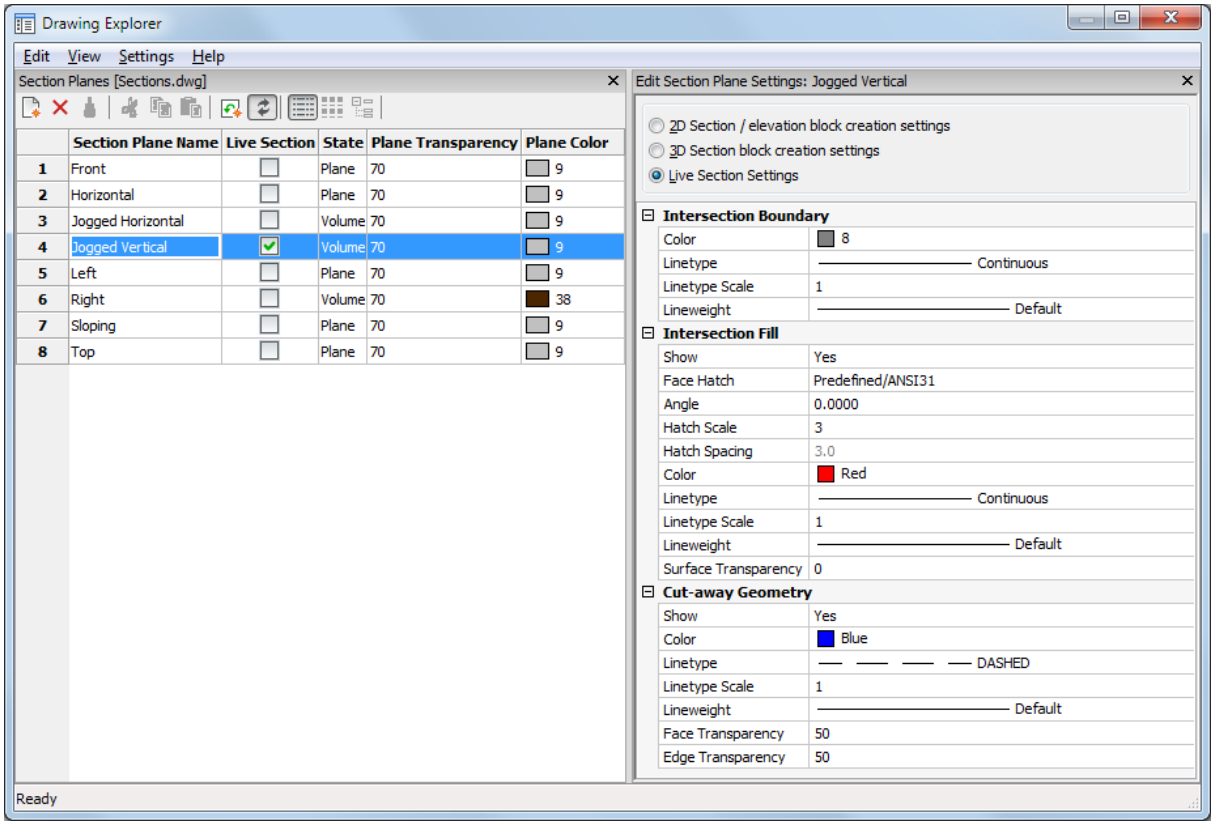
Defines the properties of a section plane entity in the *Drawing Explorer - Section Planes* dialog.

Accessing the Command

command bar: *sectionplanesettings*
menu bar: *Tools | Drawing Explorer | Section Planes*

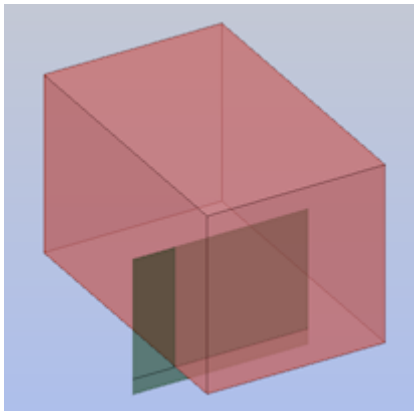
: ***sectionplanesettings***

Displays a dialog:



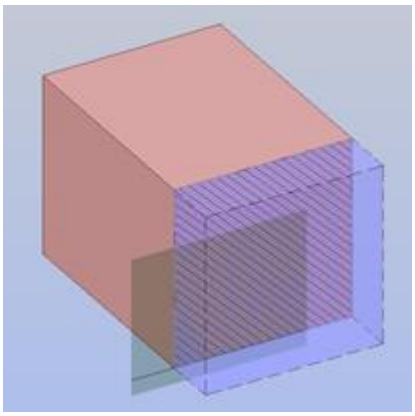
Command Options

Option	Description
	Details Panel
New	Launches the SectionPlane command to create a new section plane defintion.
Delete	Deletes the currently selected section plane definition.
Section Plane Name	Double click the field to rename the section plane.
Live Section	Click the checkbox to switch the live section property of a section plane on/off.
State	Indicates the state of the section plane. The image below shows a 3D solid (red) and the position and size of the section plane entity (green)

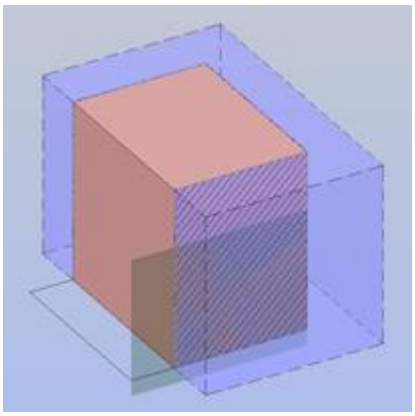


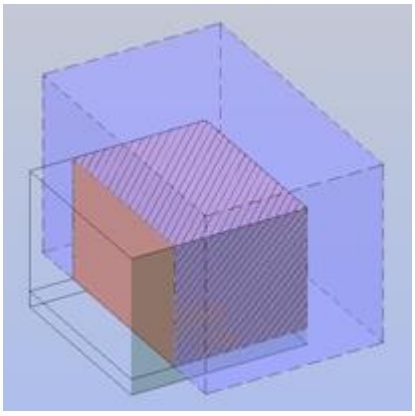
The options are:

- *Plane*: The cutting plane and the section line display.
  The cutting plane extends infinitely in all directions.
  The cut-away volume displays transparently in blue.
  If a *Plane* state is applied to a jogged section plane entity, the first segment defines the cutting plane.



- *Boundary*: The cutting plane and a 2D box, indicating the extents of the cutting plane in the XY-plane, display.
  The cutting plane extends infinitely in the Z-direction.
  The cut-away volume displays transparently in blue.



	• *Volume*: the cutting plane and a 3D box, indicating the 3D extents of the cutting plane, display. The cut-away volume displays transparently in blue. 
Plane Transparency	Sets the transparency of the cutting plane in the drawing.
Plane Color	Sets the color of the cutting plane in the drawing. The color of the section line is defined by the *Color* property of the section plane entity.
	Edit panel If you Select multiple section definitions, you can edit them simultaneously. If a setting is defined differently in the selected section definitions, the setting field reads *varies*.
2D Section / elevation block creation settings	Select multiple section plane entities to define their settings simultaneously.

Intersection Boundary	
Color	ByLayer
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Division Lines	Yes
Intersection Fill	
Show	Yes
Face Hatch	Predefined/SOLID
Angle	0.0000
Hatch Scale	1
Hatch Spacing	1.0
Color	8
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Background Lines	
Show	Yes
Hidden Line	Yes
Color	ByLayer
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Cut-away Geometry	
Show	No
Hidden Line	Yes
Color	Red
Layer	0
Linetype	DASHED
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Curve Tangency Lines	
Show	Yes
Color	Red
Layer	0
Linetype	DASHED
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer

3D Section block creation settings	Select multiple section plane entities to define their settings simultaneously.

	Intersection Boundary Color: ByLayer Layer: 0 Linetype: Continuous Linetype Scale: 1 Plot Style: ByColor Lineweight: ByLayer Show: Yes **Intersection Fill** Show: Yes Face Hatch: Predefined/SOLID Angle: 0.0000 Hatch Scale: 1 Hatch Spacing: 1.0 Color: 8 Layer: 0 Linetype: Continuous Linetype Scale: 1 Plot Style: ByColor Lineweight: ByLayer **Background Lines** Color: ByLayer Layer: 0 Linetype: Continuous Linetype Scale: 1 Plot Style: ByColor Lineweight: ByLayer **Cut-away Geometry** Show: No Color: Red Layer: 0 Linetype: DASHED Linetype Scale: 1 Plot Style: ByColor Lineweight: ByLayer
Live section settings	Select multiple section plane entities to define their settings simultaneously.

Intersection Boundary	
Color	8
Linetype	Continuous
Linetype Scale	1
Lineweight	Default
Intersection Fill	
Show	Yes
Face Hatch	Predefined/ANSI31
Angle	0.0000
Hatch Scale	1
Hatch Spacing	3.0
Color	Red
Linetype	Continuous
Linetype Scale	1
Lineweight	Default
Surface Transparency	0
Cut-away Geometry	
Show	Yes
Color	Blue
Linetype	DASHED
Linetype Scale	1
Lineweight	Default
Face Transparency	50
Edge Transparency	50

Related Commands

LiveSection - switches the *Live Section* property of a section plane on or off.
Section - creates section planes of 3D solids; the results are region entities.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneToBlock - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Explorer - Opens the Drawing Explorer dialog box.

SectionPlaneToBlock

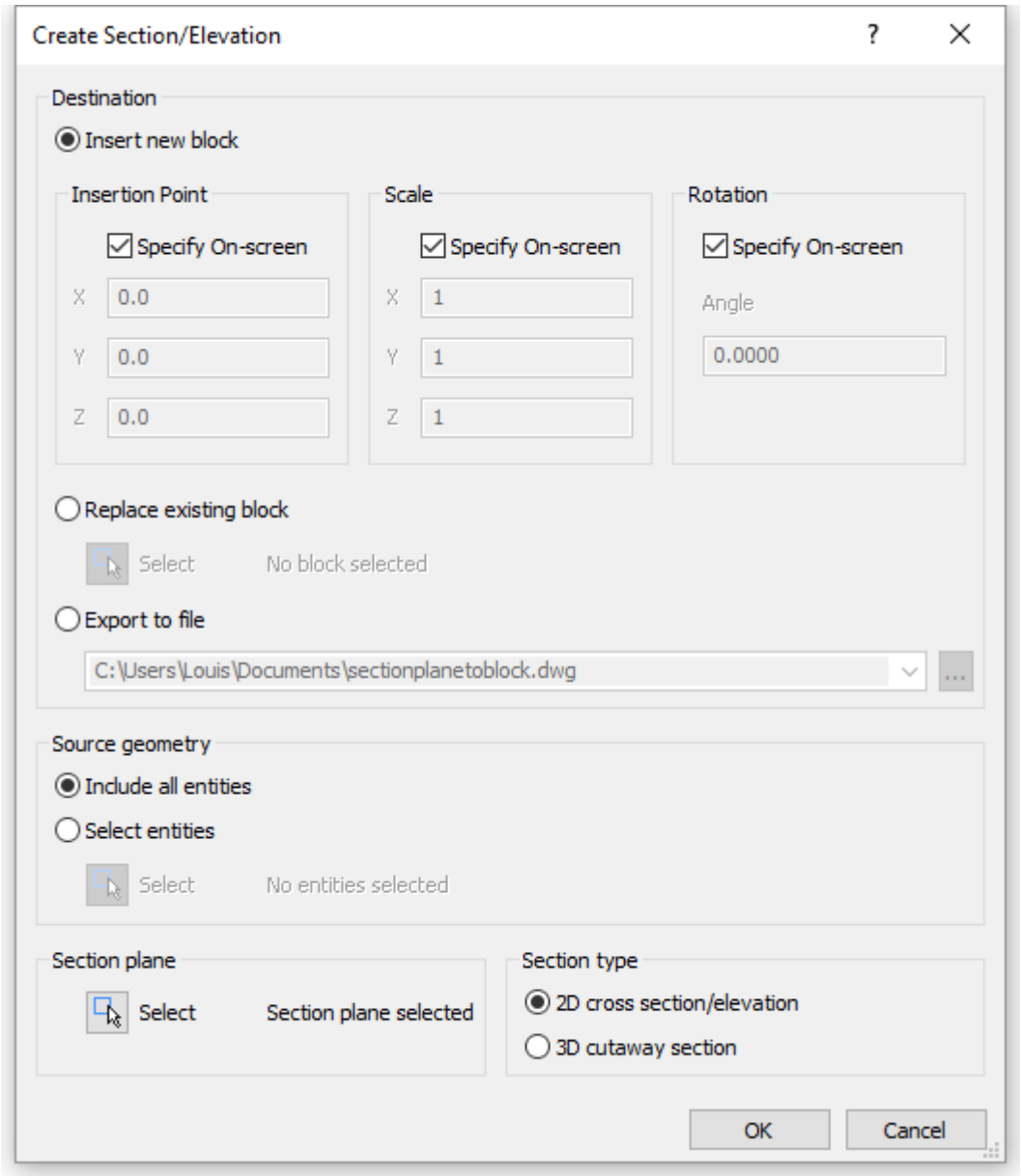
Saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Accessing the Command

command bar: *sectionplanetoblock*
menu bar: *Model | Sections | Section plane to block*
toolbar: *Sections |*

: ***sectionplanetoblock***

Displays a dialog box:



Command Options

Option	Description
Destination	Defines where the block will be placed.
Insert new block	Section geometry is saved to a new block.
Replace existing block	You are prompted to select an existing block of which the definition will be overwritten.
Export to file	Click the browse button to open the *Select Export File* dialog, then select a folder and type a name in the *File Name* field and click the *Save* button.
Source geometry	
Include all entities	All 3D geometry in the drawing will be processed.
Select entities	Click the *Select* button ( ); the *Create Section/Elevation* dialog box temporarily closes. You are prompted to select 3D entities.
Section plane	Click the *Select* button ( ); the *Create Section/Elevation* dialog box temporarily closes. You are prompted to select a section plane entity.
Section type	Defines whether a 2D or 3D block is created.
2D cross section/elevation	A 2D block is created. The 2D Section Settings of the selected section plane entity are used to create the block.
3D cutaway section.	A 3D block is created. The 3D Section Settings of the selected section plane entity are used to create the block.
Insert options	See the Insert command.
OK	The *Create Section/Elevation* dialog box closes. In case a new block is created, you are prompted to specify the insertion point, scale and rotation.
Cancel	Aborts the command.

Procedures

To insert a section as a 2D block
To insert a section as a 3D block
To replace an existing block
To export a section to a file

Related Commands

LiveSection - switches the *Live Section* property of a section plane on or off.
Section - creates section planes of 3D solids; the results are region entities.
SectionPlane - creates a section entity that creates sections of 3D solids.
SectionPlaneSettings - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.

Security

See VbaSecurity

SecurityOptions

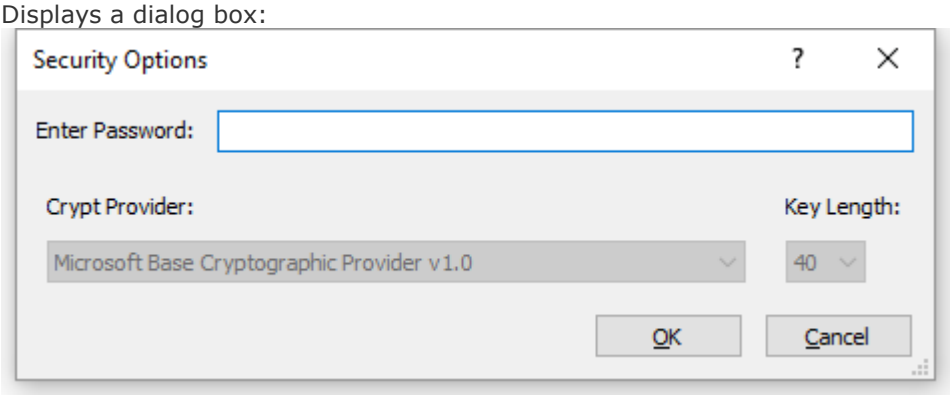
Sets a password to protect your drawing.

Accessing the Command

command bar: *securityoptions*
alias: none
menu bar: *File | Security Options*

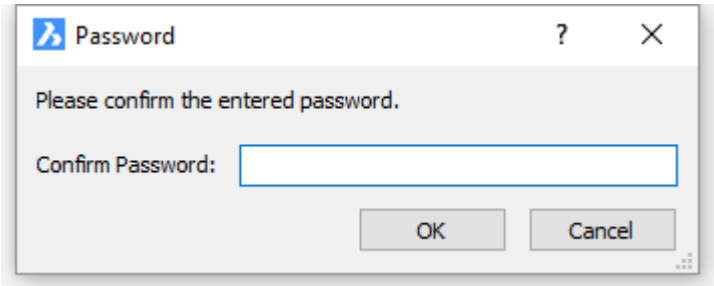
: ***securityoptions***

Displays a dialog box:



Key-in the password, then click the OK button.

A dialog box displays:



Confirm the password, then click the OK button.

Command Options

Option	Description
Crypt Provider	Specifies the encryption level for drawing properties. Choose an encryption provider supplied the operating system. Microsoft Base Cryptographic Provider v1.0 Microsoft Base DSS and Diffie-Hellman Cryptographic Provider Microsoft Base Smart Card Crypto Provider Microsoft DH SChannel Cryptographic Provider Microsoft Enhanced Cryptographic Provider v1.0 Microsoft Enhanced DSS and Diffie-Hellman Cryptographic Provider Microsoft Enhanced RSA and AES Cryptographic Provider Microsoft RSA SChannel Cryptographic Provider Microsoft Strong Cryptographic Provider
Key Length	Sets the length of the encryption key. The longer the key, the higher the level of protection is. 40 48 56

Procedures

To set the password
To remove the password

Related Commands

Dwgprops: opens the Drawing Properties dialog box
QSave - saves the drawing without displaying the Save dialog box.
Save - saves the current drawing through a dialog box.
SaveAll - saves all open drawings.
Open: opens a drawing
WhoHas - reports the ownership of a drawing file.

Select

Selects one or more entities; can be accessed later with the Previous option.

Accessing the Command

command bar: *select*
shortcut: *Ctrl+A selects all entities on non-frozen layers*

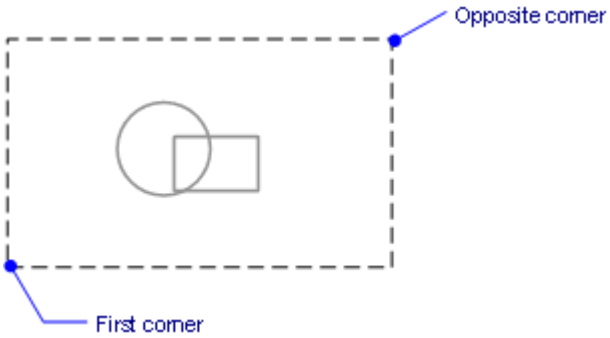
: ***select***
Prompts you in the command bar:

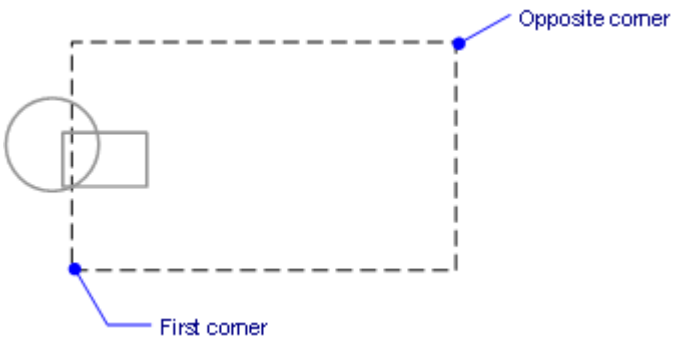
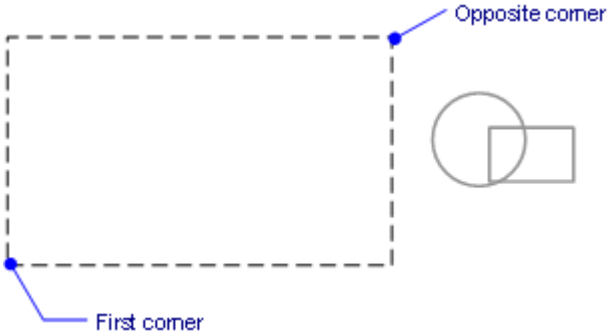
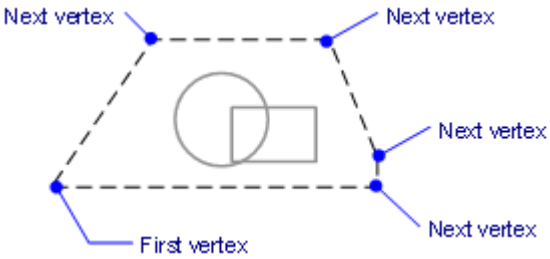
Select entities to include in set: (Choose one or more entities using any selection mode, listed below.)

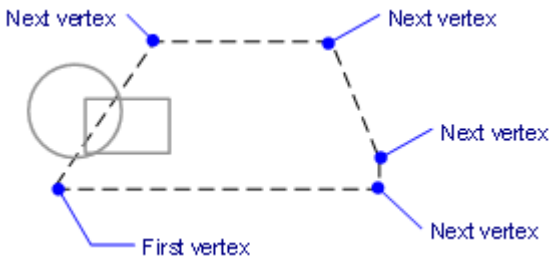
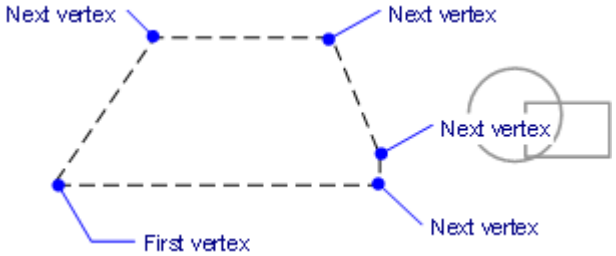
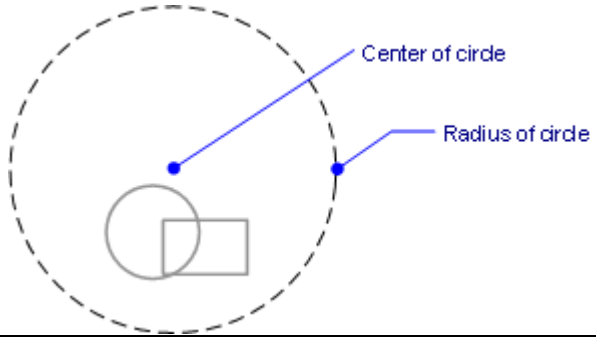
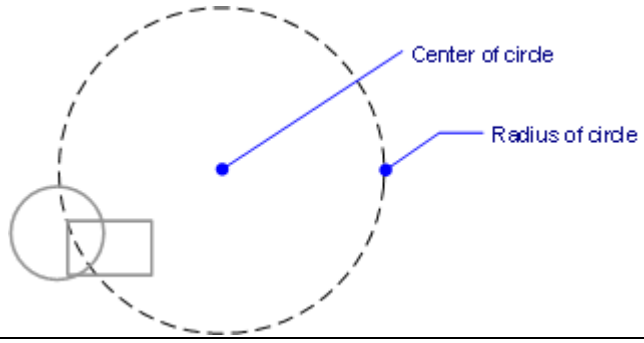
Select entities to include in set: (Choose more entities, or press Enter to exit command.)

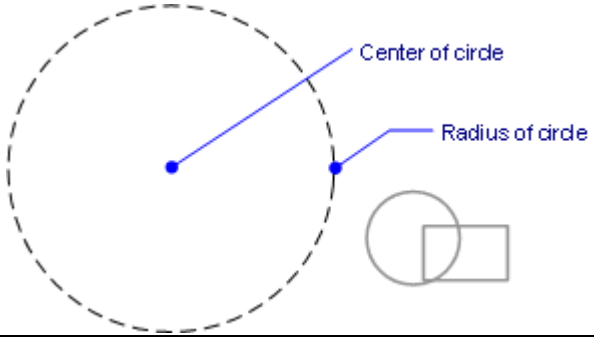
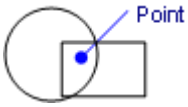
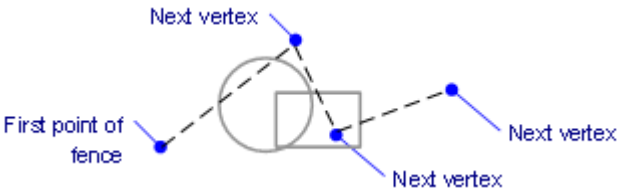
You can use any selection mode (listed below) at the 'Select entities' prompt found in many editing commands.

Command Options

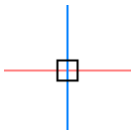
Option	Abbreviation	Description
Select entities to include in set		Chooses entities. You can use any of the following selection options:
All	ALL	Selects all entities in the drawing, except for those on frozen layers.
Window	W	Selects all entities fully inside a rectangular selection window; prompts you: First corner of inclusive window - specify the first point of the selection rectangle. Opposite corner - specify the other corner. 
Crossing	C	Selects all entities inside and crossing a rectangular selection window; prompts you: First corner of crossing window - specify the first point of the selection rectangle. Opposite corner - specify the other corner.

		
Outside	O	Selects all entities fully outside a rectangular selection window; prompts you: First corner of exclusive window - specify the first point of the selection rectangle. Opposite corner - specify the other corner. 
Window polygon	WP	Selects all entities fully inside a polygonal selection window; prompts you: First vertex of inclusive polygon - specify the first point of the selection polygon. Next vertex - specify the next point. Next vertex - press Enter to close the vertex. 
Crossing polygon	CP	Selects all entities inside and crossing a polygonal selection window; prompts you: First vertex of crossing polygon - specify the first point of the selection polygon. Next vertex - specify the next point. Next vertex - press Enter to close the vertex.

		
Outside polygon	OP	Selects all entities fully outside a polygonal selection window; prompts you: First vertex of outside polygon - specify the first point of the selection polygon. Next vertex - specify the next point. Next vertex - press Enter to close the vertex. 
Window circle	WC	Selects all entities fully inside a circular selection window; prompts you: Center of circle - pick a center point. Radius of circle - specify a radius by picking or entering a value 
Crossing circle	CC	Selects all entities inside and crossing a circular selection window; prompts you: Center of circle - pick a point. Radius of circle - specify a radius by picking or entering a value. 

Outside circle	OC	Selects all entities fully outside a circular selection window; prompts you: Center of circle - pick a point. Radius of circle - specify a radius by picking or entering a value. 
Point	PO	Selects all closed entities surrounding the pick point; prompts you: Point to locate entity around - specify a point. 
Fence	F	Selects all entities that cross the fence selection lines; prompts you: First point of fence - specify the starting point of the fence. Next vertex - specify the next point. Next vertex - press Enter to finish drawing the fence. 
Previous	P	Reuses the entities selected with the previous selection.
Last	L	Selects the last entity drawn.
Add	A	Adds additional entities to the selection set; prompts you: Select entities to include in set - add entities to the selection set. This option is typically used following the Subtract option.
Subtract	R	Removes entities from the selection set; prompts you: Subtract entities from selection set - remove entities from the selection set.
Undo	U	Undoes the last action during this command.
Group	G	Selects a group; Prompts you: Enter group name - type the name of a group.
Settings	D	Displays the *Selection* section of the Settings dialog box:

<table>
<tr>
<td></td>
<td></td>
<td>
<table>
<tr><td colspan="2">⊟ Selection</td></tr>
<tr><td>Pick add</td><td>☑ Add selected entities to current selection set</td></tr>
<tr><td>⊞ Pick automatic</td><td>0x0003 (3)</td></tr>
<tr><td>Pick box</td><td>4</td></tr>
<tr><td>Pick drag</td><td>[1] Draw selection window using dragging</td></tr>
<tr><td>Pick first</td><td>[1] First select entities, then issue a command</td></tr>
<tr><td>⊟ Pick style</td><td>0x0001 (1)</td></tr>
<tr><td>1</td><td>☑ Group selection</td></tr>
<tr><td>2</td><td>☐ Associative hatch selection (Not yet supported)</td></tr>
</table>

PICKAUTO: Controls automatic inside or crossing window selection. Set the value to 3 (default) to start window selection when the cursor is over an entity. If set to 1, the entity under the cursor is selected.
PICKBOX: Sets the size in pixels of the small square attached to the crosshairs. Entities highlight when when hit by the pickbox.

PICKDRAG: Controls the method for window selection:
• 0: Click to start, then select a second point.
• 1: Press and hold the left mouse button, then move the mouse and release the mouse button.
PICKFIRST: Controls whether commands accept an existing selection set.
• 0: Existing selection sets are neglected when a command is launched.
• 1: Commands which prompt to select entities use an existing selection set.
PICKSTYLE: Controls the selection of groups and associative hatches. The value is stored as a bitcode using the sum of the values of the selected options. .
• 0: No group or associative hatch selection.
• 1: Allows the selection of groups.
• 2: Allows the selection of associative hatches (not yet supported).
• 3: Allows the selection of groups and associative hatches (not yet supported).
Ctrl+H toggles the value of PICKSTYLE
</td>
</tr>
<tr>
<td>Properties</td>
<td>PRO</td>
<td>Selects entities on the basis of their properties; prompts you:
Select by properties - enter a property mode.
See Properties command.</td>
</tr>
<tr>
<td>Color</td>
<td>C</td>
<td>Selects all entities of the same color; prompts you:
Color to select - specify a color number or name.
See Color command.</td>
</tr>
<tr>
<td>Layer</td>
<td>LA</td>
<td>Selects all entities of the same layer names; prompts you:
Layer name(s) to select - specify one or more layer names.
Use commas to separate multiple layer names.</td>
</tr>
</table>

		See Layer command.
Linetype	LI	Selects all entities of the same linetype; prompts you: Linetype(s) to select - specify one or more linetype names. Use commas to separate multiple linetype names. See Linetype command.
Name	N	Selects all entities of the same entity name; prompts you: Entity name(s) to select - specify one or more entity names. Use commas to separate multiple entity names. Entity names are the names of entities, such as line, circle, arc, and ray.
Thickness	TH	Selects all entities of the same thickness; prompts you: Thickness to select - enter a number. See Elev command.
Value	V	Selects all entities with the same value; prompts you: Value(s) to select - enter one or more values. Use commas to separate multiple entity values.
Width	W	Selects all entities of the same width; prompts you: Width to select - enter a number.
Handle	H	Select the entity with the handle number; prompts you: Entity handle to select - enter a number, such as 889. Handles are hexadecimal numbers that uniquely identify every entity in drawings. To determine the handle number, use the List command.
Location	LO	Returns to the original prompt: **Select entities to include in set** - add entities to the selection set.

Procedures

Selection and grip settings
Composing a selection set before (pre-pick)
Composing a selection set afterwards (post-pick)
Selecting overlapping entities
To create a selection set using QuickSelect

Related Commands

DdSelect - sets selection properties in the *Entity Selection* section of the *Settings* dialog box.
SelectSimilar - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE system variable.
dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelGrips - selects entities and displays grips.
QSelect - composes a selection set using filters.

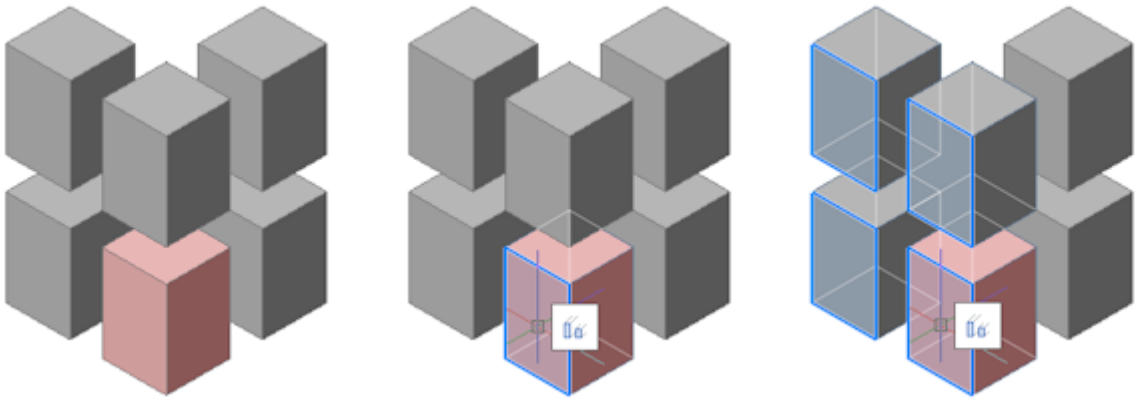
SelectAlignedFaces

Selects all faces in a model which are coplanar with a selected face.

Accessing the Command

command bar: *selectalignedfaces*
Quad cursor menu: *Solid/Face Selection* |

: ***selectalignedfaces***
Prompts you in the command bar:
Select Face: (Click a face in the model.)



Start | Select face | Aligned faces selected

Command Options

No options.

Related Commands

BimMultiSelect - selects linear solids with coplanar and/or parallel axes.
dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelectAlignedSolids - selects all solids in a model of which a face is coplanar with a selected face.
SelectConnectedFaces - selects all faces in a model which are connected to a selected face.
SelectConnectedSolids - selects all solids in a model which are connected to a selected face.

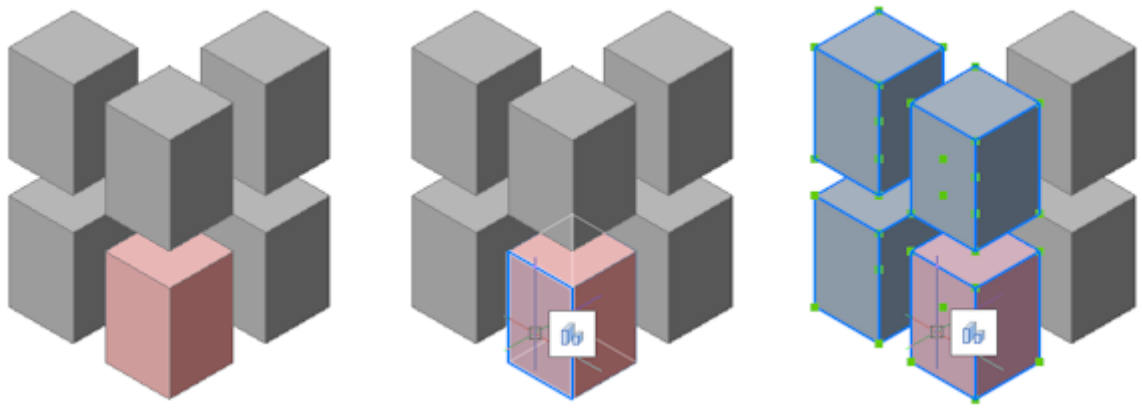
SelectAlignedSolids

Selects all solids in a model of which a face is coplanar with a selected face.

Accessing the Command

command bar: *selectalignedsolids*
Quad cursor menu: *Solid/Face Selection* |

: ***selectalignedsolids***
Prompts you in the command bar:
Select Face: (Click a face in the model.)



Start | Select face | Aligned solids selected

Command Options

No options.

Related Commands

BimMultiSelect - selects linear solids with coplanar and/or parallel axes.
dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelectAlignedFaces - selects all faces in a model which are coplanar with a selected face.
SelectConnectedFaces - selects all faces in a model which are connected to a selected face.
SelectConnectedSolids - selects all solids in a model which are connected to a selected face.

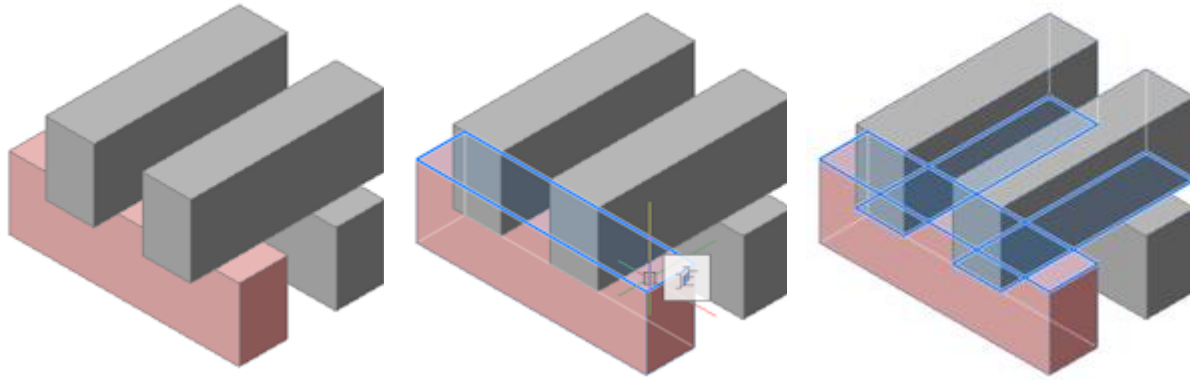
SelectConnectedFaces

Selects all faces in a model which are connected to a selected face.

Accessing the Command

command bar: *selectconnectedfaces*
Quad cursor menu: *Solid/Face Selection* |

: ***selectconnectedfaces***
Prompts you in the command bar:
Select Face: (Click a face in the model.)



Start | Select face | Connected faces selected

Command Options

No options.

Related Commands

BimMultiSelect - selects linear solids with coplanar and/or parallel axes.
dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelectAlignedFaces - selects all faces in a model which are coplanar with a selected face.
SelectAlignedSolids - selects all solids in a model of which a face is coplanar with a selected face.
SelectConnectedSolids - selects all solids in a model which are connected to a selected face.

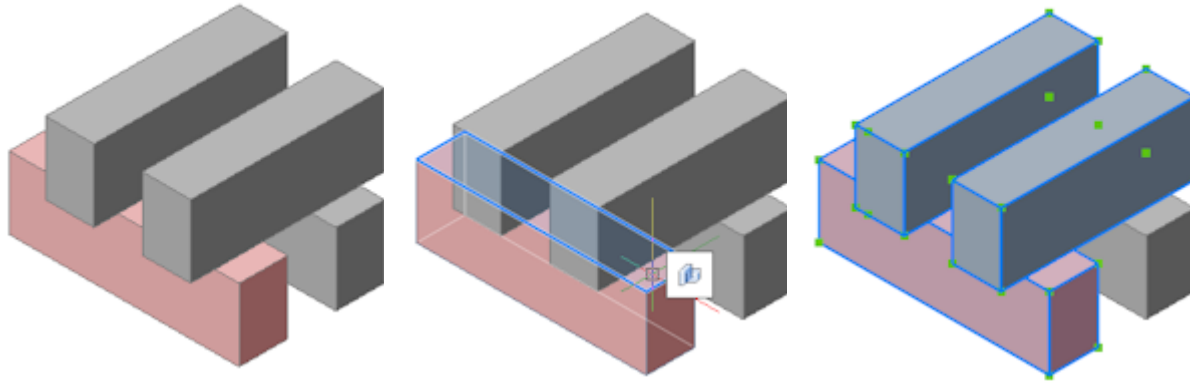
SelectConnectedSolids

Selects all solids in a model which are connected to a selected face.

Accessing the Command

command bar: *selectconnectedsolids*
Quad cursor menu: *Solid/Face Selection* |

: ***selectconnectedsolids***
Prompts you in the command bar:
Select Face: (Click a face in the model.)



Start | Select face | Connected solids selected

Command Options

No options.

Related Commands

BimMultiSelect - selects linear solids with coplanar and/or parallel axes.
dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SelectAlignedFaces - selects all faces in a model which are coplanar with a selected face.
SelectAlignedSolids - selects all solids in a model of which a face is coplanar with a selected face.
SelectConnectedFaces - selects all faces in a model which are connected to a selected face.

SelectSimilar

Selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE system variable.

Accessing the Command

command bar: *selectsimilar*
menu bar: *Edit | Select Similar*
toolbar: *Standard |*

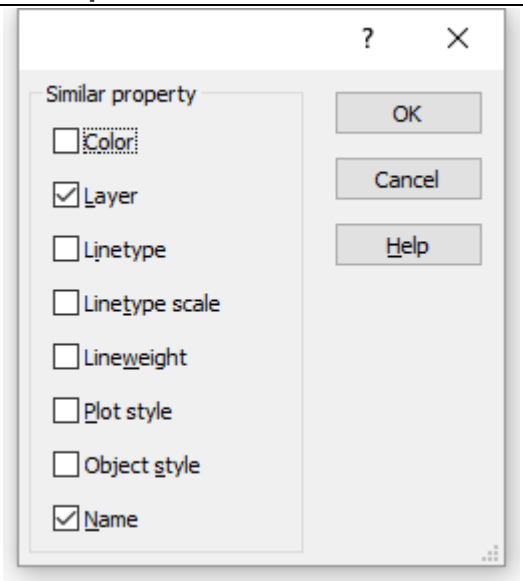
: ***selectsimilar***
Prompts you in the command bar:

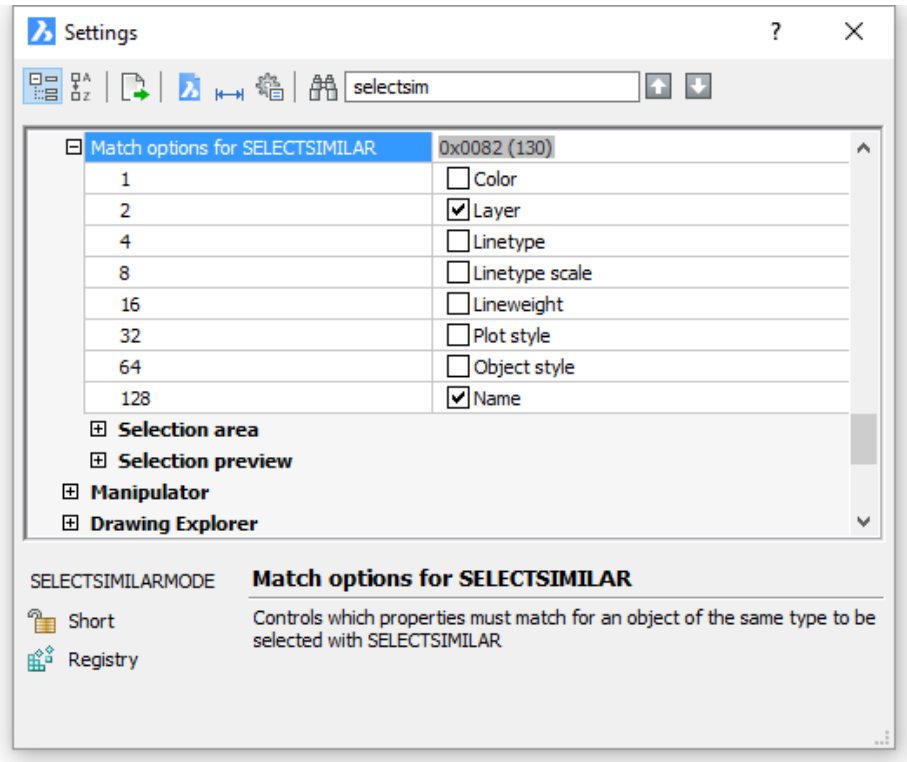
Select entities or [SEttings]: (Select one or more entities or type SE to display a dialog box to edit the SELECTSIMILARMODE system variable.)
Entities in set: 1

Select entities or [SEttings]: (Select more entities or right click to select similar entities.)

All entities that match the current selection and the current value of the SELECTSIMILARMODE system variable are selected.

Command Options

Option	Description
Settings	 Check the properties to be taken into account when searching for similar entities. e.g. check *Color* to select all lines of the same color as the selected line. Alternatively set the value of the SELECTSIMILARMODE system variable in the SETTINGS dialog/



Related Commands

QSelect - composes a selection set using filters.
Select - selects one or more entities; can be accessed later with the Previous option.
SelGrips - selects entities and displays grips.

SelGrips

Selects entities and displays grips (short for "select grips").

Accessing the Command

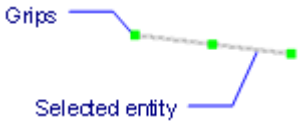
command bar: *selgrips*
shortcut: *Click entities with the cursor.*
alias: *selgrip*

: ***selgrip***
Prompts you in the command bar:

Select entities to display grips: (Choose one or more entities.)

Select entities to display grips: (Choose more entities, or press Enter to end the command.)

BricsCAD highlights selected entities and displays their grips.



You can edit the entities by dragging the grips.

To select all entities in the drawing, use Ctrl+A.

Command Options

Option	Description
Select entities to display grips	Chooses one or more entities.

Related Commands

Select - selects entities.
SelectSimilar - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE system variable
DdGrips - sets the parameters for grips through the Settings dialog box.

Settings

Changes the values of system variables and user preferences through a dialog box.

Accessing the Command

command bar: *settings*
aliases: *ddrmodes, rm, options, dsettings, ddsetvar*
menu bar: *Settings | Settings*
toolbar: *Settings |*

: ***settings***
Displays the Settings dialog box.

Related Commands

Options - sets program options through the Settings dialog box.
SetVar - changes system variables through the command bar.
SettingsSearch - opens the Settings dialog box at the specified category, system variable or user preference.

SettingsSearch

Opens the Settings dialog box at the specified category, system variable or user preference.

Accessing the Command

command bar: *settingssearch*

: ***settingssearch***
Prompts you in the command bar:
Search setting/category: (type the name of a setting category, system variable or user preference).
Please notice that the search is case sensitive: the names of system variables are fully capitalized (e.g. INSUNITS), user preferences are mixed (e.g. SnapMarkerSize).

Command Options

No options

Related Commands

Settings and DdSetVar - changes the values of system variables through a dialog box.
SetVar - changes system variables through the command bar.

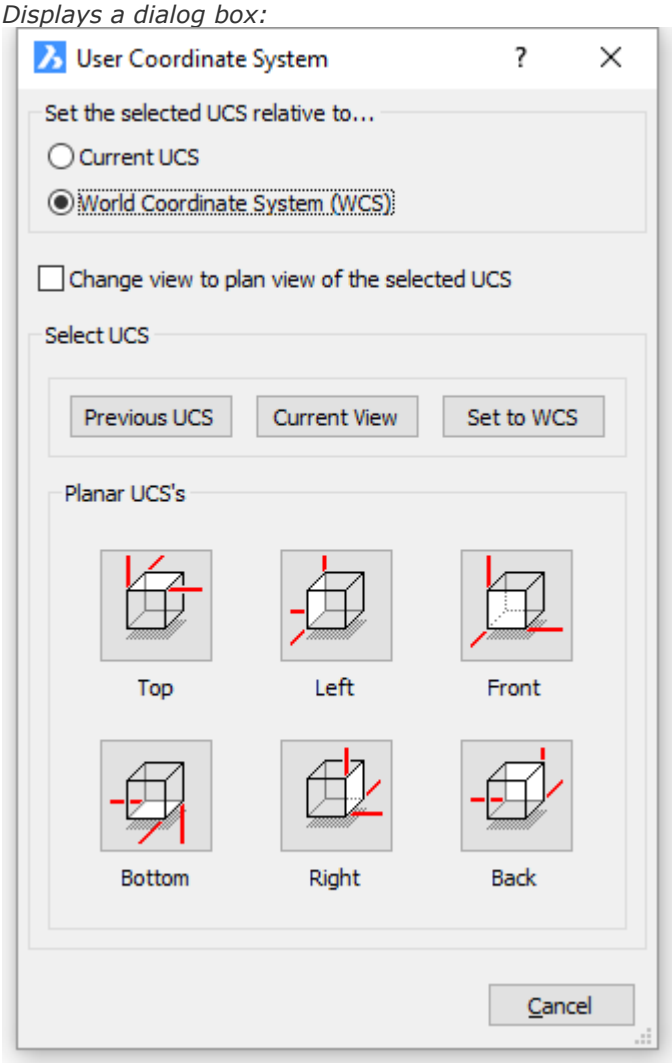
SetUcs

Sets the UCS to a specified viewpoint (short for "set user-defined coordinate system").

Accessing the Command

command bar: *setucs*
aliases: *dducsp, ucp*
menu bar: *Settings | Set UCS*
toolbar: *Settings |*

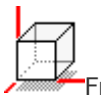
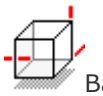
: ***setucs***
Displays a dialog box:



Click on a button; the dialog box disappears automatically.

*Use the **UCS** command's View option to name the UCS generated by this command.*

Command Options

Option	Description
Set the selected UCS relative to	Specifies how the new UCS is set:
Current UCS	THe UCS is set relative to the current UCS.
World Coordinate System	The UCS is set relative to the WCS. WCS is short for "world coordinate system." The WCS is the default coordinate system when no UCS is set; it cannot be changed.
Change view to plan view of the selected UCS	Toggles whether the plan view is displayed when switching UCSs: Yes - plan view is displayed. No - the viewpoint does not change. Plan view looks down the z axis at the x,y plane.
Select UCS	Sets the UCS to one of the following coordinate systems. Clicking one of these buttons sets the UCS, and then immediately exits the dialog box.
Previous UCS	Changes the UCS to the previous UCS.
Current View	Changes the UCS to match the current viewpoint.
Set to WCS	Changes the UCS to match the WCS.
Planar UCSs	Sets the UCS to one of the following standard views. Clicking one of these buttons sets the UCS, and then immediately exits the dialog box.
Top	Changes the UCS to match the top view.
Left	Changes the UCS to match the left view.
 Front	Changes the UCS to match the front view.
Bottom	Changes the UCS to match the bottom view.
Right	Changes the UCS to match the right view.
 Back	Changes the UCS to match the back view.
Cancel	Closes the dialog box without changing the UCS.

Related Commands

Ucs - creates and modifies user-defined coordinate systems.
UcsIcon - controls the appearance of the UCS icon.
ExpUcs - creates, modifies, and deletes named UCSs through the Drawing Explorer.
Plan - changes the viewpoint to the plan view of the WCS or current UCS.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
Settings - sets parameters of UCSs.

SetVar

Displays and changes the values of system variables (short for "set variables").

Accessing the Command

command bar: *setvar*
transparent: *'setvar*
alias: *set*
menu bar: *Tools | Set Variable*

: ***setvar***
Prompts you in the command bar:
Variable name or ?: (Enter the name of a system variable, or type ?.)

For example:

Variable name or ?: chamferd

New current angle for CHAMFERD <0>: (Enter a value, or press Enter to accept the current value.)

This command is unnecessary, because system variables can be entered directly at the command prompt:

: chamferd

New current angle for CHAMFERD <0>: (Enter a value, or press Enter to accept the current value.)

You can also change the values of system variables with the Settings command.

The values of read-only variables cannot be changed. For example:

: program

PROGRAM = "BRICSCAD" (read only)

Command Options

Option	Description
Variable name	Specifies the name of a system variable.
?	Lists the names of system variables; prompts you: Variable(s) to list <*> - enter one of the following: • * to list all system variables. • name* to list all system variables starting with name. • name to list one system variable.

Related Commands

Settings and DdSetVar - change the values of system variables through a dialog box.
SettingsSearch - opens the Settings dialog box at the specified category, system variable or user preference.

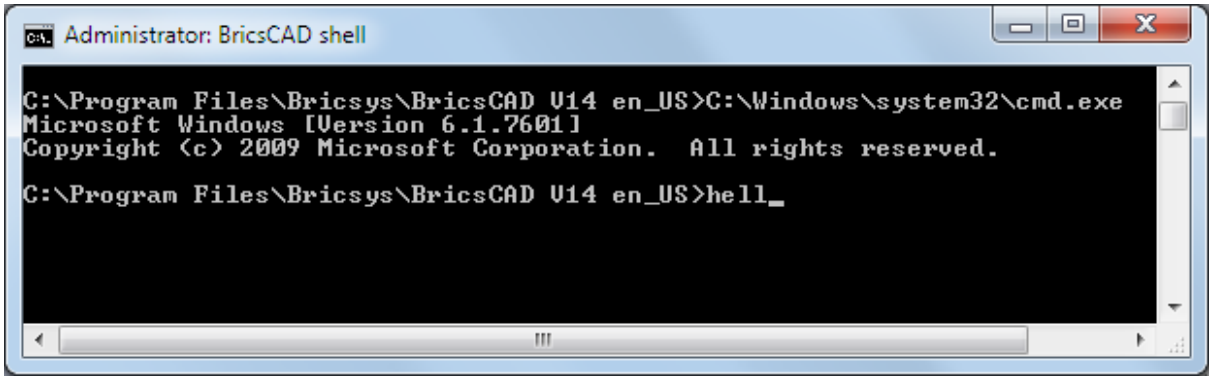
Sh

Opens Windows command prompt window; runs other applications (short for "shell").

Accessing the Command

command bar: *sh*
transparent: *'sh*
alias: *shell*

: ***sh***
Prompts you in the command bar:
OS Command: (Enter the name of a Windows program, or press Enter for the command prompt window.)



Enter commands or names of programs, such as 'Notepad'.

For a list of commands, enter Help.

To exit the window, enter Exit.

Command Options

Option	Description
OS Command	Specifies the name of the Windows program to run. Press Enter to open the Windows command prompt window.

Related Commands

Start - starts other applications.
Shell - opens the operating system command prompt window; runs other applications.
Files - displays the operating system's File Manager.

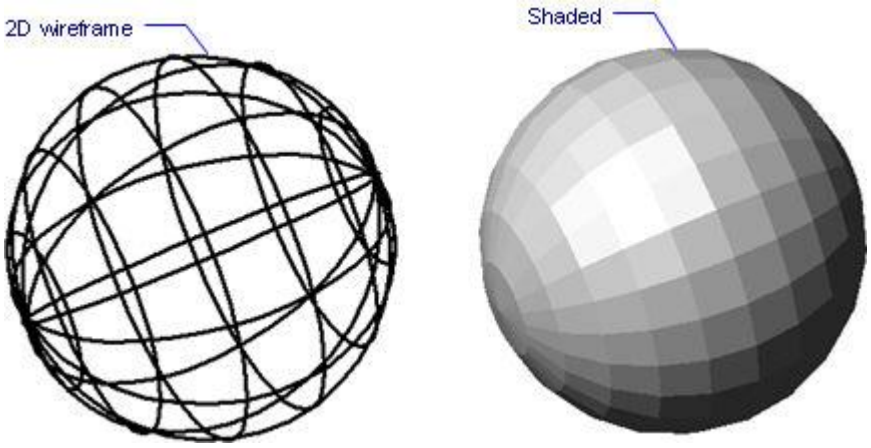
Shade

Generates shaded images of 3D drawings.

Accessing the Command

command bar: *shade*
alias: *sha*
menu bar: *View | Rendering | Shade*
toolbar: *Rendering |*

: ***shade***
Prompts you in the command bar:
In order to switch back to wireframe mode run command SHADEMODE



To return to wireframe mode, use the ***ShadeMode*** *command's 2D option.*

Command Options

No options.

Related Commands

Hide - removes hidden lines from 3D entities.
Render - generates a photorealistic rendering using materials and lights
ShadeMode - sets the shading style for 3D entities.

ShadeMode

Sets the shading style for 3D drawings.

Accessing the Command

command bar: *shademode*
alias: *vscurrent*
toolbar: *Render* |
properties bar: *Visual Style*

: ***shademode***
Prompts you in the command bar:

Current mode: 2D wireframe

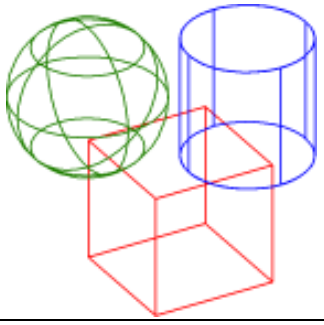
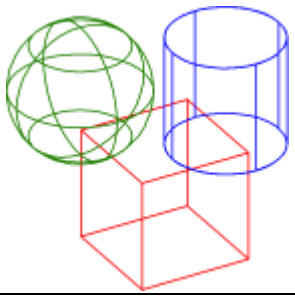
Enter option [2D wireframe/3D wireframe/Hidden/Flat/Gouraud/Flat+edges/gOuraud+edges/Realistic/Modeling] <2D wireframe>:
(Enter an option.)

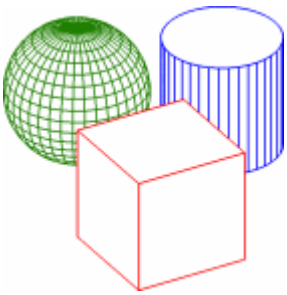
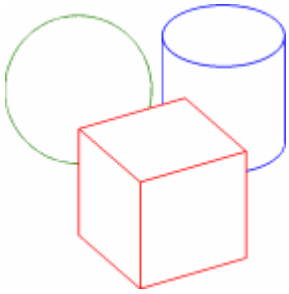
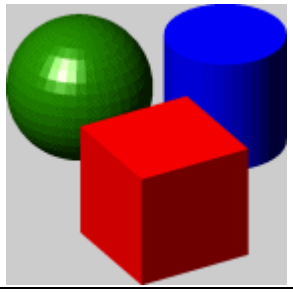
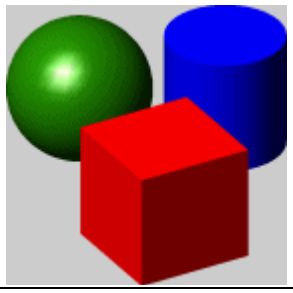
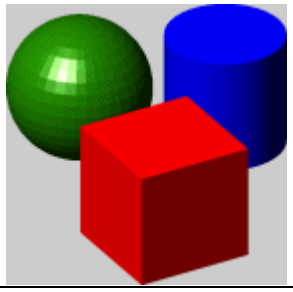
The shade mode is applied to the drawing immediately.

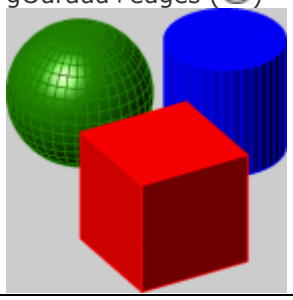
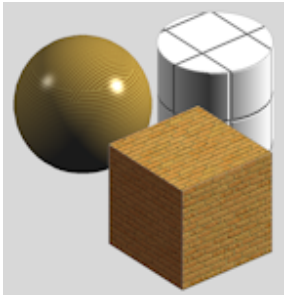
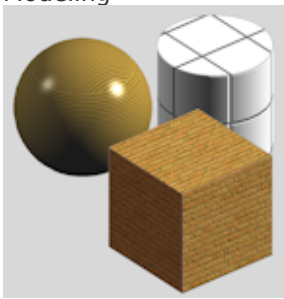
You can draw and edit in any of these shade modes.

For more realistic renderings, use the Render command (Pro and Platinum version only).

Command Options

Option	Description
2D wireframe ( ) 	Displays drawings in 2D wireframe mode. This is the normal mode for viewing drawings in BricsCAD.
3D wireframe ( ) 	Displays drawings in 3D wireframe mode. This mode may display certain items differently from 2D wireframe mode.
Hidden ( )	Displays drawings in hidden mode. Hidden lines are removed. If the DISPSILH system variable is on, silhouette lines of curved surfaces display only; if DISPSILH is off segment lines display on curved surfaces.

 *display silhouette lines off*  *display silhouette lines on*	In 2D wireframe mode, zooming or panning restores the wireframe view. If 3D wireframe mode, hidden mode is saved when zooming or panning.
Flat ( ) 	Displays drawings in flat rendered mode. Facets are not smoothed. This is the mode used by the **Shade** command.
Gouraud ( ) 	Displays drawings in Gouraud rendered mode. Facets are smoothed.
Flat+edges ( ) 	Displays drawings in flat rendered mode, with edges outlined in the background color.

gOuraud+edges ( ) 	Displays drawings in Gouraud rendered mode, with edges outlined in the background color.
Realistic 	Displays drawings using the *Realistic* visual style.
Modeling 	Displays drawings using the *Modeling* visual style.

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. Note: high anti-alias values incur a high calculation cost.

ANTIALIASSCREEN = 1

ANTIALIASSCREEN = 4

Related Commands

Hide - removes hidden lines from 3D entities.
Render - generates a photorealistic rendering using materials and lights
RedSdkInfo - displays rendering related hardware and driver specifications in the command bar.
Shade - generates shaded images of 3D drawings.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

Shape

Places shapes in drawings.

Accessing the Command

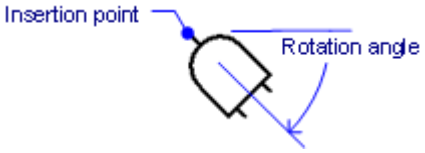
command bar: *shape*

: ***shape***
Prompts you in the command bar:
? to list/Shape to insert: *(Enter the name of a shape, or type?)*

Insertion point for shape: (Pick a point or enter x,y coordinates.)

Scale factor for shape <1.0>: (Enter a positive or negative number.)

Rotation angle for shape <0>: (Enter a positive or negative number.)



Before you can place shapes in drawings, you must first load a SHP shape file with the **Load** command.

Shapes are an early version of blocks that were highly efficient but difficult to code; shapes are rarely used anymore.

The SHP format is also used by certain fonts; it contains a flag to distinguish between shapes and fonts.

To code shapes, see *Customizing BricsCAD*.

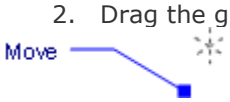
Command Options

Option	Description
Shape to insert	Specifies the name of the shape; prompts you: Insertion point for shape - specify the insertion point. Pick a point, or enter x,y coordinates. Scale factor for shape - specify the scale factor. Enter a positive or negative number larger or smaller than 1.0 Negative numbers mirror the shape. Rotation angle for shape - specify the rotation angle. Enter a positive or negative number. Positive angles are measured clockwise from the positive x axis.
? to list	Lists the names of shapes loaded in the current drawing. Reports the file name and names of shapes. If no shapes are loaded, prompts you: *** no file *** Use the Load command to load shapes.

Grips Editing

Shapes can be edited directly through grips:

1. Select the shape. Notice that it has one grip at its insertion point.
2. Drag the grip to move the shape.



Related Commands

Load - loads SHP shape files.
Insert - inserts blocks into drawings.

SheetSet

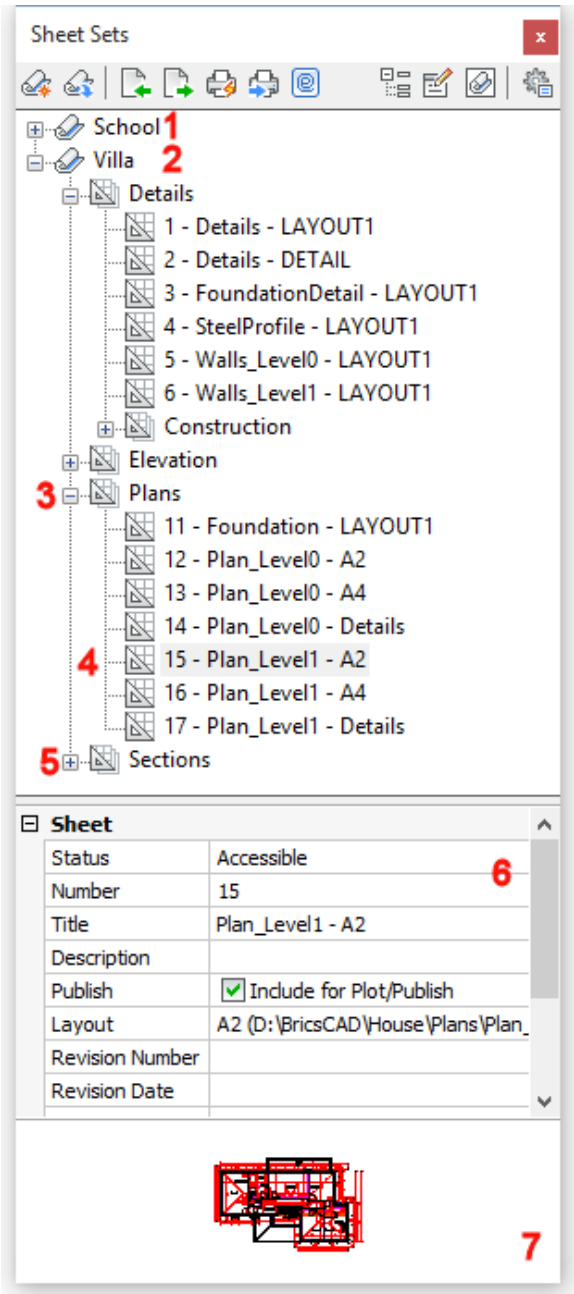
Opens the Sheet Sets panel.

Accessing the Command

command bar: *sheetset*
menu bar: *Tools | Sheet Sets...*
toolbar: *Standard |*
context menu: *Sheet Sets*

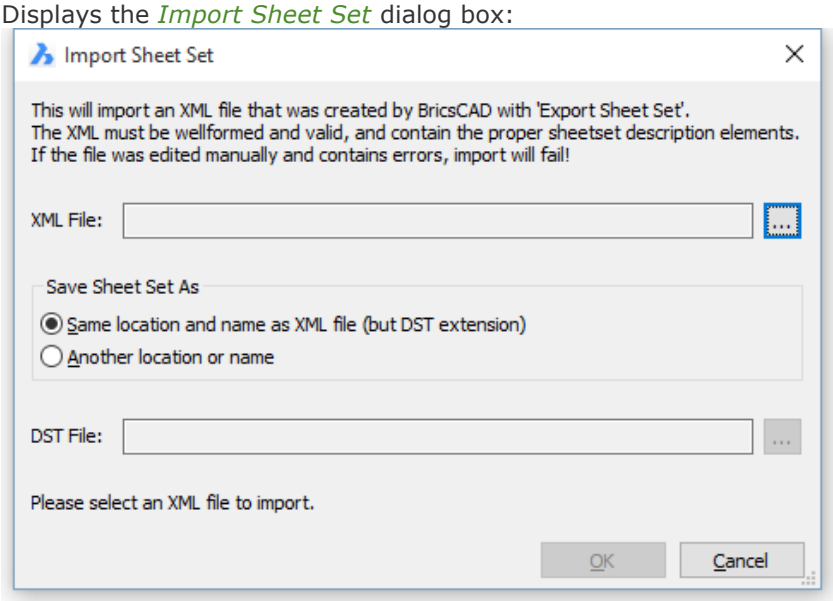
: ***sheetset***

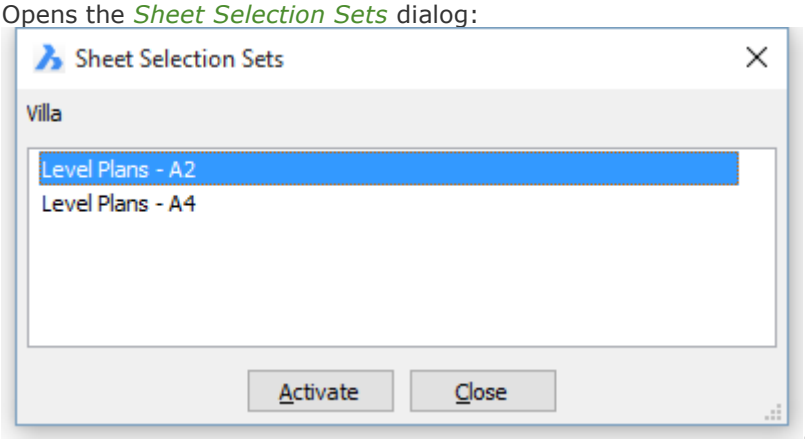
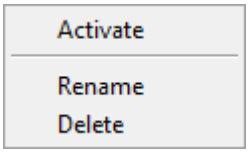
Displays a panel:

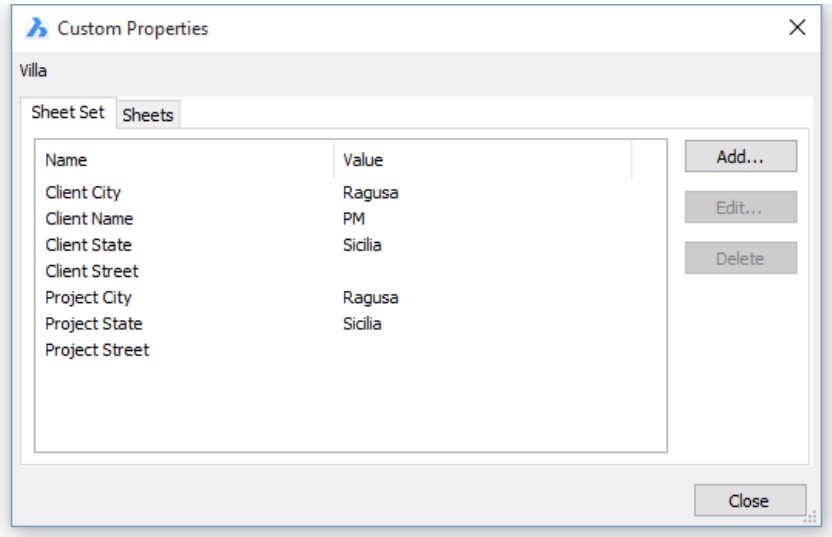
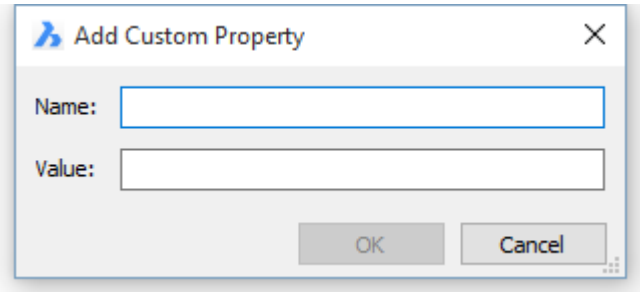
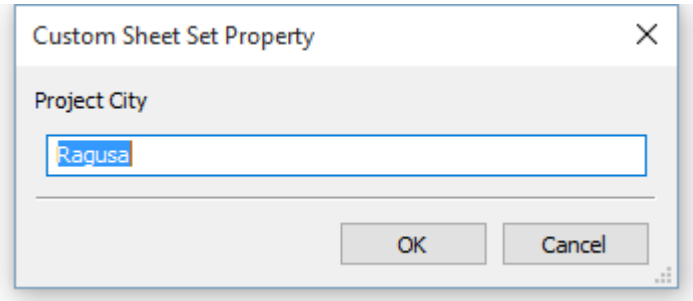


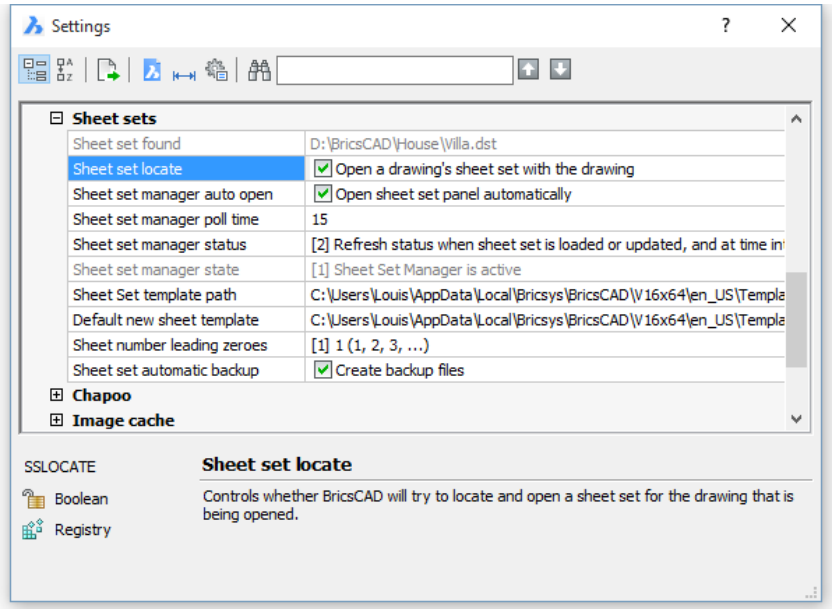
1. Sheet Set (collapsed)
2. Sheet Set (expanded)
3. Sheet Subset (Expanded)
4. Sheet
5. Sheet Subset (collapsed)
6. Properties and variables of the currently selected sheet set, subset or sheet
7. Preview of the currently selected sheet

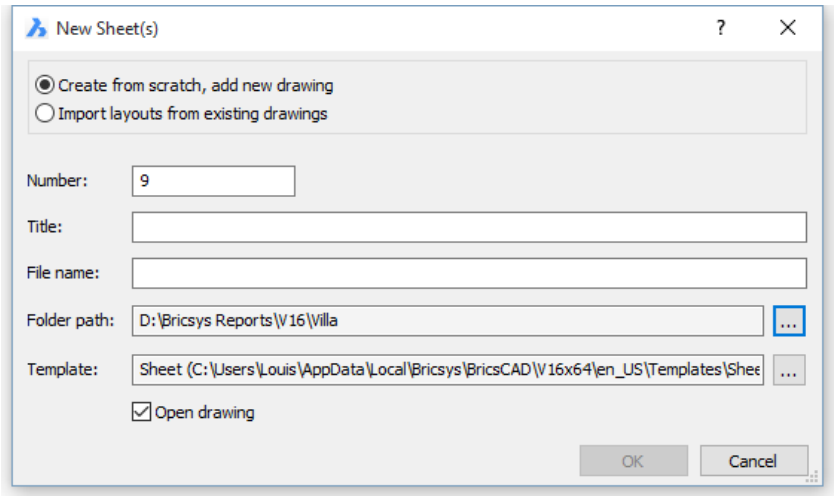
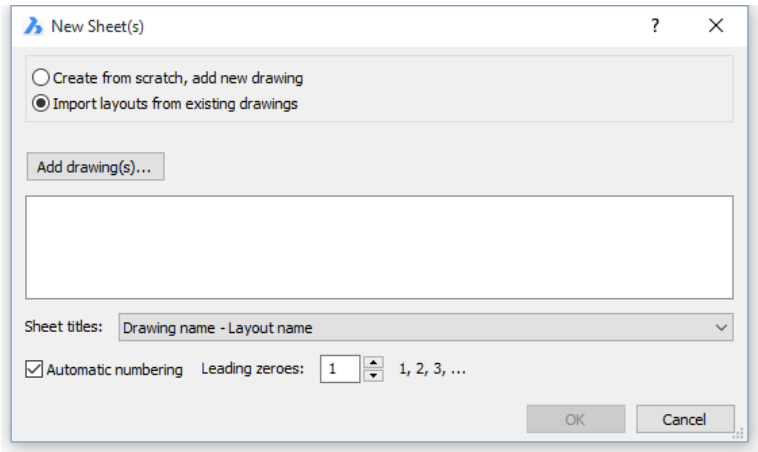
Command Options

Option	Description
New sheet set	Launches the NewSheetSet command.
Open sheet set	Launches the OpenSheetSet command. When a DST file is dropped on the application, it is opened in the Sheet Set Explorer.
Export to XML	Exports a sheet set in XML format. Displays the *Export Sheet Set* dialog box.
Import from XML	Displays the *Import Sheet Set* dialog box:  • *XML File*: Click the *Browse* (...) button, then select an XML file in the *Select XML File* dialog box. • *Save Sheet Set As*: Choose whether to save the imported XML file as DST file using the same location and name as the XML file or to save it to a new location or name. If you choose the latter option, click the *Browse* button (...) at the right hand side of the *DST File* field to specify a folder and file name. • *OK*: Imports the XML file and saves it to a DST file.
Plot	Plots the currently selected sheet set(s), subset(s) or sheet(s). Depending on the *Publish Sheets in Subset* property of a subset and the *Publish* property of sheets, sheets might not be included.
Publish	Publishes the currently selected sheet set(s), subset(s) or sheet(s). Depending on the *Publish Sheets in Subset* property of a subset and the *Publish* property of sheets, sheets might not be included.

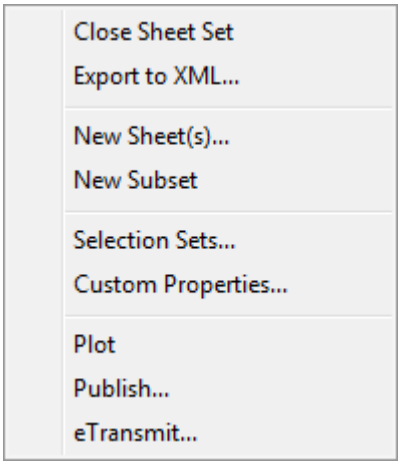
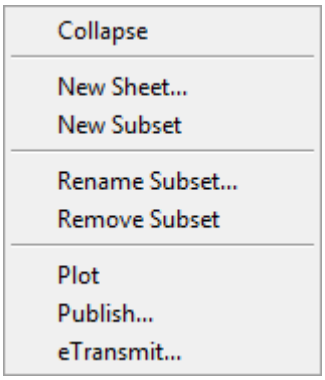
eTransmit	Launches the eTransmit command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure.
Selection sets	Opens the *Sheet Selection Sets* dialog:  • Select a selection set, then click the *Activate* button to restore the selection. • Right click a selection set to display a context menu: 

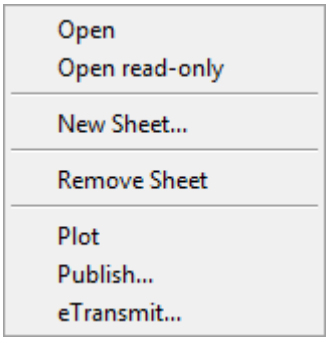
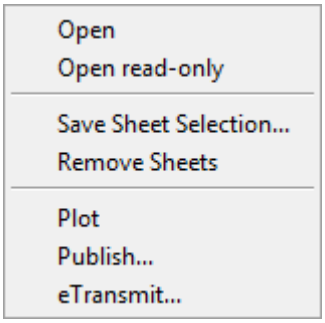
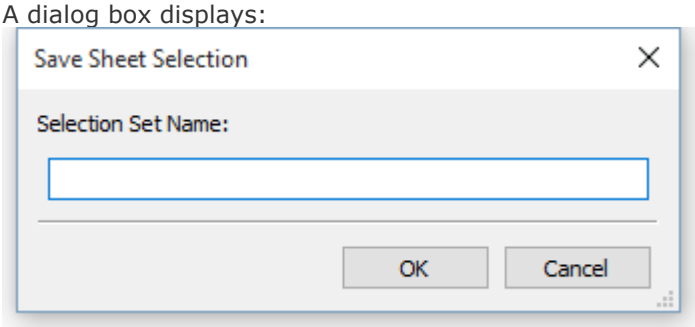
Custom Properties	Opens the *Custom Properties* dialog, which allows to add custom sheet set and sheet properties. Alternatively right click the sheet set name or a sheet name in the sheets tree, then choose *Custom Properties...* in the context menu.  • Click the *Sheet Set* tab to manage custom sheet set properties. • Click the *Sheets* tab to manage custom sheet properties. • Click the *Add* button to add a custom property. Displays the Add Custom Property dialog box:  • Select a custom property, then click the *Edit* button to edit the value of a custom property. Displays the *Custom Sheet Property* or *Custom Sheet Set Property* dialog.  • Select a custom property, then click the *Delete* button to delete the selected property.

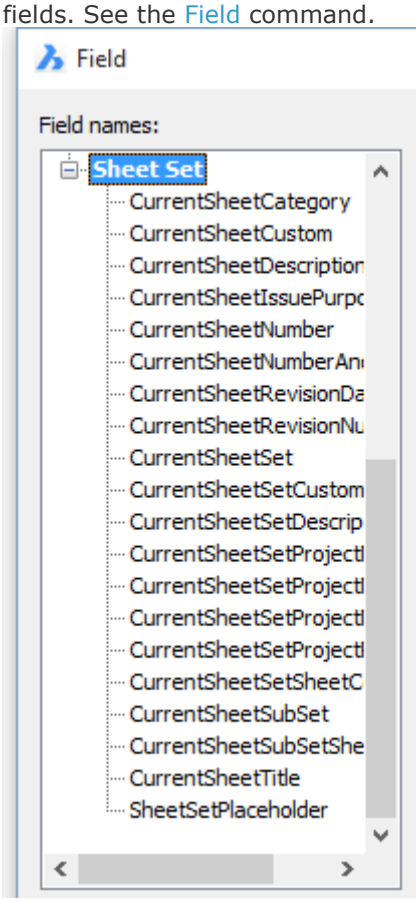
View Categories	Allows to expand/collapse the currently loaded sheet sets
Sheet Set Options	Opens the *Sheet Sets* options in the Settings dialog.  Sheet set found (SSFOUND): Displays the sheet set file name associated with the current drawing. Sheet set locate (SSLOCATE): Controles whether BricsCAD tries to locate and open a sheet set when a drawing is being opened. Sheet set manager auto open (SSMAUTOOPEN): Controls whether or not BricsCAD displays the Sheet Set Manager when a drawing is opened which is associated with a Sheet Set. Sheet set manager poll time (SSMPOLLTIME): Controls the time interval between automatic refereshes of the status data in a sheet set if the value of SSMSHEETSTATUS = 2. Sheet set manager status (SSMSHEETSTATUS): Controls whether the status data in a sheet set is refreshed. Sheet set manager state (SSMSTATE): Determines whether the Sheet Set Manager is active or not. Sheet set template path (SheetSetTemplatePath): Specifies the path to the Sheet Set Templates folder. Click the *Browse* button (...), then select the path in the *File* dialog. The default path is: *C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD\Vxx\en_US\Templates\Sheet Sets* Default new sheet template (DefaultNewSheetTemplate): Specifies the default drawing template file (*.dwg or *.dwt) for new sheets. Click the *Browse* button (...), then select the file in the *File* dialog. Sheet number leading zeroes (SheetNumberLeadingZeroes): Specifies how to format the *Number* value of new sheets. Sheet set automatic backup (SheetSetAutoBackup): Determines whether a backup is created whenever a sheet set is opened. The backup files have a *.ds$ extension.

New Sheet(s)	Right click the sheet set name, a subset or a sheet, then choose *New Sheet(s)...* in the context menu. The *New Sheet(s)* dialog box displays:  • The *Create from scratch, add new drawing* option, creates a new drawing and adds a layout in the drawing to the sheet list in the current sheet set or subset. *Number*: Accept the automatically calculated number or type a new number. *Title*: Type a name in the Title field *File name*: Accept the automatically generated name (<Number>+<Title>) or type a new name. *Folder Path*: Accept the current path or press the *Browse* button (...) to set a different path. *Template*: Accept the current template or press the *Browse* button (...) to select a different template drawing. *Open drawing*: If checked, the newly created drawing opens when you click the *OK* button. • The *Import layouts from existing drawings* creates new sheets. The *New Sheet(s)* dialog changes: 

	Click the *Add drawing(s)...* button, then select the drawings,you want to import layouts from, in the *Select Drawings* dialog. The layout(s) in the selected drawing(s) are listed: Uncheck the layouts you do not want to import as a sheet. Choose a option from the *Sheet titles* list button. The options are: *Drawing name*, *Drawing name - Layout name*, *Layout name*. Choose whether or not to number the new sheets automatically. Press the *OK* button to create the new sheet(s).

Context Menus	
 	Right click a sheet set to open the *Sheet Set* context menu. • *Close Sheet Set*: Closes the sheet set. • *Export to XML*: Displays the *Export Sheet Set As* file dialog box. Writes the sheet set definition to an XML file. • *New Sheet...*: Displays the New Sheet dialog box, which allows to add new sheet(s) to the current sheet set. • *New Subset*: Adds a new subset to the current sheet set. • *Selection Sets...*: Displays the Selection Sets dialog box. • *Custom Properties...*: Displays the Custom Properties dialog box. • *Plot*: Prints all sheets in the sheet set. • *Publish*: Launches the Publish command and loads all sheets in the sheet set and its subsets. • *eTransmit...*: Launches the eTransmit command.
 	Right click a sheet subset to open the *Subset* context menu. • *Collapse* / *Expand*: Collapses or expands the subset. • *New Sheet...*: Displays the New Sheet dialog box, which allows to add a new sheet to the subset. • *New Subset*: Adds a new subset to the subset. • *Rename Subset*: Allows to enter a new name for the subset. • *Remove Subset*: Deletes the subset. The deletion is definitive when the modifications are saved when closing the sheet set. • *Plot*: Prints all sheets in the subset. • *Publish*: Launches the Publish command and loads all sheets in the subset. • *eTransmit...*: Launches the eTransmit command.

	Right click a sheet to open the *Sheet* context menu. • *Open*: Opens the drawing with the layout of the selected sheet. • *Open read-only*: Opens the drawing with the layout of the selected sheet in read-only mode. • *New Sheet...*: Displays the New Sheet dialog box, which allows to add a new sheet to the sheet set or current subset. • *Remove Sheet*: Deletes the sheet. The deletion is definitive when the modifications are saved when closing the sheet set. • *Plot*: Prints the sheet. • *Publish*: Launches the Publish command and loads the sheet. • *eTransmit...*: Launches the eTransmit command.
	Right click when multiple sheets are selected to open the *Sheet Selection* context menu. Press and hold the Ctrl key to select multiple sheets. Sheets in different subsets of the same sheet set can be selected. Choose *Save Sheet Selection...* to save the currently selected sheets. A dialog box displays:  Type a name for the sheet selection, then click the *OK* button.
Mouse actions	
Double click	Expands/Collapses a sheet set or a subset. Opens a sheet.

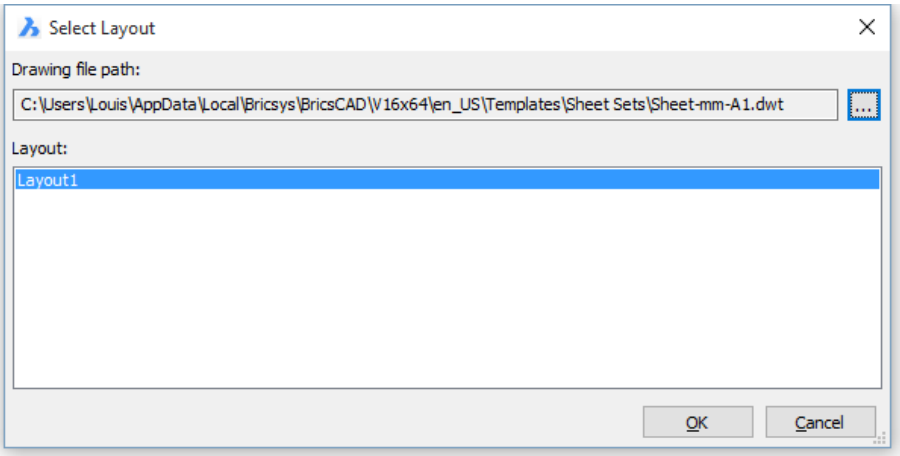
Define Properties	
	Sheet set properties, subset properties and sheet properties can be used in fields. See the Field command. 

Sheet Set properties

Sheet Set	
Name	FamilyHouse
Description	House in Brighton project
File Path	C:\BricsCAD\House\House.dst
Total Sheets	23
Project Control	
Project Number	
Project Name	
Project Phase	
Project Milestone	
Sheet Creation	
New Sheet Location	C:\BricsCAD\House
Sheet Template Layout	A4_Level0 (C:\Bricscad Training\Bricscad\TrainingCD\EN\Exercises\House.dwg)
Sheet Set Custom Properties	
Address	5 Sea Drive, Brighton
Architect	Michael Brickstone
Owner	Mr. Fisherman
Sheet Custom Properties	
Size	A0

- Select a field, then define the property.
- *New Sheet Location*: Click the *Browse* button (...) then specify a folder.

Sheet Template Layout: Click the *Browse* button (...), then choose a template drawing and a layout.



- *Prompt*: Choose whether or not to be prompted for a template drawing when creating a new sheet from scratch.

<table>
<tr>
<td>Subset properties</td>
<td>
<table>
<tr><td colspan="2">⊟ Subset</td></tr>
<tr><td>Name</td><td>Construction</td></tr>
<tr><td>Description</td><td></td></tr>
<tr><td>Publish Sheets in Subset</td><td>Publish by sheet 'Include for publish' setting</td></tr>
<tr><td>New Sheet Location</td><td>C:\BricsCAD\House</td></tr>
<tr><td>Sheet Template Layout</td><td>A4_Level0 (C:\Bricscad Training\Bricscad\TrainingCD\EN\Exercises\House.dwg)</td></tr>
<tr><td>Total Sheets</td><td>3</td></tr>
</table>

Name and description: Select the field, then type a name and description.
Publish Sheets in Subset: Choose an option.
New Sheet Location: Click the Browse button (...) then specify a folder.
Sheet Template Layout: Click the Browse button (...), then choose a template drawing and a layout.

</td>
</tr>
<tr>
<td>Sheet properties</td>
<td>
<table>
<tr><td colspan="2">⊟ Sheet</td></tr>
<tr><td>Number</td><td>6</td></tr>
<tr><td>Title</td><td>Walls_Level1 - LAYOUT1</td></tr>
<tr><td>Description</td><td></td></tr>
<tr><td>Publish</td><td>☑ Include for Plot/Publish</td></tr>
<tr><td>Layout</td><td>LAYOUT1 (C:\BricsCAD\House\Details\Walls_Level1.dwg)</td></tr>
<tr><td>Revision Number</td><td></td></tr>
<tr><td>Revision Date</td><td></td></tr>
<tr><td>Issue Purpose</td><td></td></tr>
<tr><td>Category</td><td></td></tr>
<tr><td colspan="2">⊟ Sheet Custom Properties</td></tr>
<tr><td>Size</td><td>A0</td></tr>
</table>

Number, Title, Description, Revision Number, Revision Date, Issue Purpose and Category: Select a field, then define the property. The Title field is mandatory, the other fields are optional.
Publish: Choose whether or not to include the sheet when the sheet set or the subset is published or plotted.
Layout: Click the Browse button (...) then specify a different drawing and/or layout.
Sheet Custom Properties: Specify the custom properties for the sheet.

</td>
</tr>
</table>

Procedures

Using sheet set properties in a title block
To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

SheetSetHide - closes the Sheet Set panel.
NewSheetSet - creates a new sheet set.
OpenSheetSet - opens an existing sheet set; displays a file dialog box.
-OpenSheetSet - opens an existing sheet set; prompts you in the command bar.
Publish - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

-Publish - prints a saved sheet list (= a list of model space or paper space layouts).

-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

SheetSetHide

Closes the Sheet Set panel.

Accessing the Command

command bar: *sheetsethide*

: ***sheetsethide***
No prompt

Command Options

No command options

Related Commands

SheetSet - manages sheet sets. Opens the Sheet Sets panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Shell

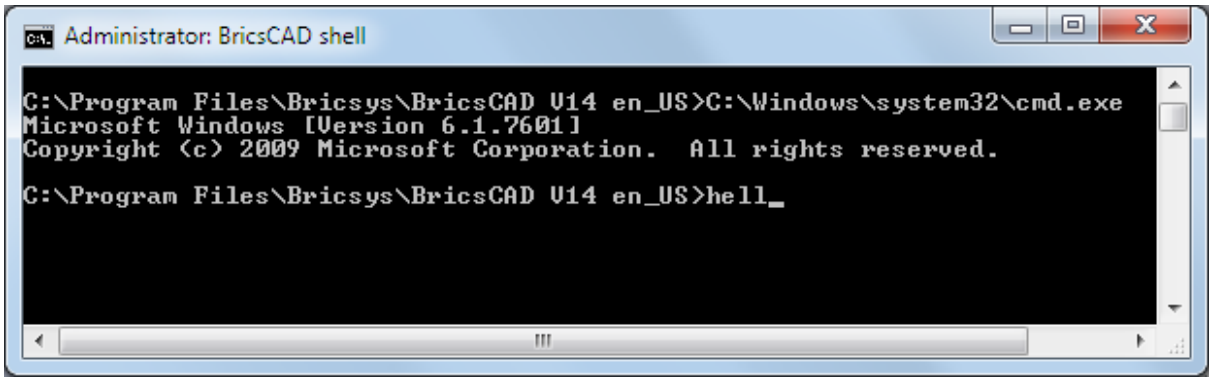
Opens Windows command prompt window; runs other applications.

Accessing the Command

command bar: *shell*
transparent: *'shell*
*a*lias: *sh*

: ***shell***

Prompts you in the command bar:
OS Command: *(Enter the name of a Windows program, or press Enter for the command prompt window.)*



Enter commands or names of programs, such as 'Notepad'.

For a list of operating system commands, enter Help.

To exit the window, enter Exit.

Command Options

Option	Description
OS Command	Specifies the name of the Windows program to run. Press Enter to open the Windows command prompt window.

Related Commands

Start - starts other applications.
Sh - opens the operating system command prompt window; runs other applications.
Files - displays the operating system's File Manager.

Singleton

Determines whether multiple copies of BricsCAD can run at the same time.

Accessing the Command

command bar: *singleton*
transparent: *'singleton*

: ***singleton***
Prompts you in the command bar:

Allow only single BricsCAD application instance? [Yes/No] <No>: (Type Y or N.)

When set to no, only a single instance of BricsCAD can run.

When set to yes, you can launch two or more copies of BricsCAD at the same time.

NOTES

- Some third-party applications run only in a single instance of BricsCAD.
- To run multiple instances of BricsCAD in the *Metro* interface of *Windows 8*, you must right click the BricsCAD shortcut icon, then choose *Open in a new window* in the context menu.

Command Options

Option	Description
Allow only single BricsCAD application instance	Toggles single instance mode of BricsCAD: • Yes - allows only one copy of BricsCAD to run. • No - allows two or more copies of BricsCAD to run.

Related Commands

None.

Sketch

Creates freehand sketches.

Accessing the Command

command bar: *sketch*
alias: *freehand*
menu bar: *Draw | Sketch*
toolbar: *Draw |*

: ***sketch***
Prompts you in the command bar:

Sketch: Length of segments <0.1000>: (Enter a number, identify two points or press Enter to accept the default.)

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

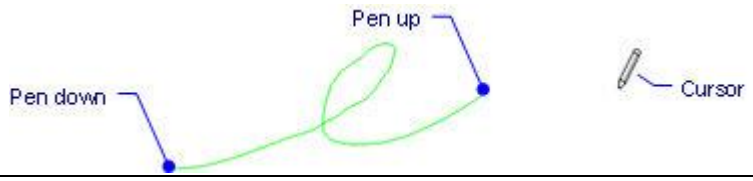
Click to start sketching, click again to toggle pen up/down.

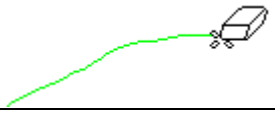
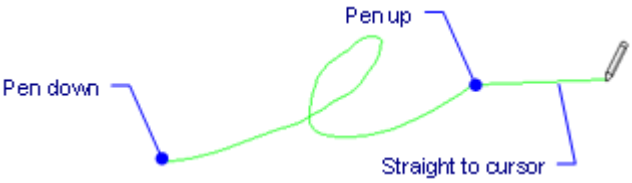
Enter an option, or press X followed by Enter to exit command.
During sketching, the lines are green until the sketch lines are saved:

Saving sketch lines ...

Depending of the status of the SKPOLY system variable sketches are made from lines or polylines.

Command Options

Option	Description
Length of segments	Specifies the length of segments. Enter a number. Too small a segment length results in large drawings.
Toggle Pen	Allows you to move the pen without drawing. 
Quit	Exits the command without saving the sketch.
Record	Saves the sketch; reports: Saving sketch lines. **Continue sketching.**

Erase	Erases the sketch lines from their endpoints interactively. Click to stop erasing and restart sketching. 
Connect	Connects cursor to end of the last segment; prompts you: **Point cursor to end of last sketch line** - move the cursor to the end of the last sketch. This option is meant for use after a pen up.
Straight	This option works only when the pen is up. Draws a straight segment to the current cursor location . 
Exit with save	Saves the sketch; reports: Saving sketch lines. **Exits the sketch command.**

Procedures

Setting the Sketch Poly (SKPOLY) system variable
Creating a freehand sketch

Related Commands

PLine - draws polyline segments and arcs.
RevCloud - draws revision clouds.

Slice

Cuts a 3D solid with a plane; the result is a 3D solid cut in two.

Accessing the Command

command bar: *slice*
alias: *sl*
menu bar: *Model | 3D soldis| Slice*
toolbar: *3D Solids |*

: ***slice***
Prompts you in the command bar:

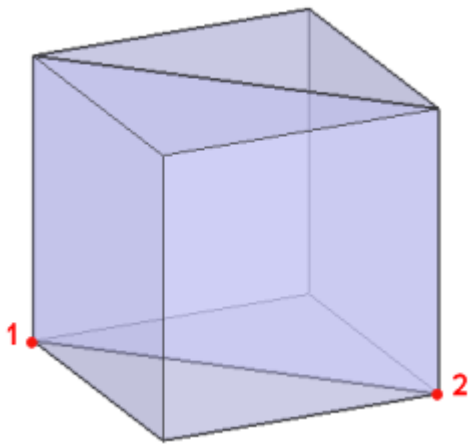
Select entities: (Choose one or more 3D solids.)
Entities in set: 1

Select entities: (Choose more 3D solids, or press Enter to continue.)

Specify first point on slicing plane or [Object/Surface/Zaxis/View/XY/YZ/ZX/Line-point/3points] <3points>: (Pick a point (1) or enter an option.)

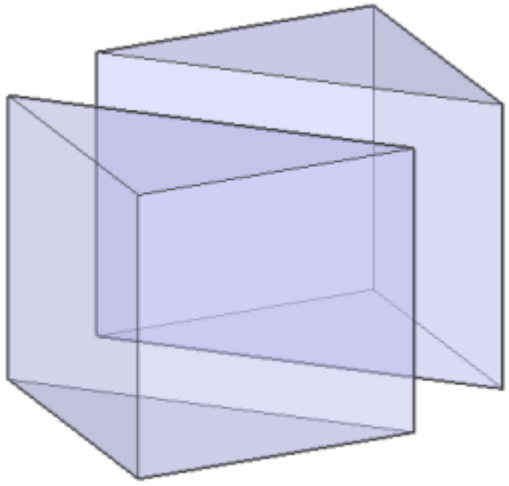
Specify second point on slicing plane: (Pick a point (2), or enter x,y,z coordinates.)

Specify a point on desired side of the plane or Both: (Choose one of the parts; the other one is erased. Or type B to retain both.)

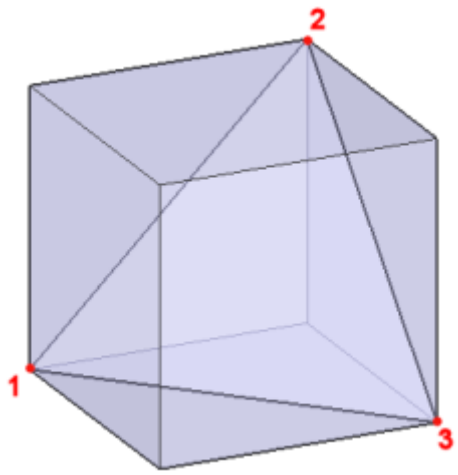


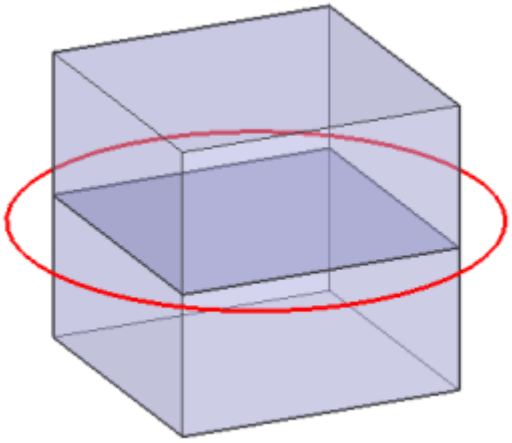
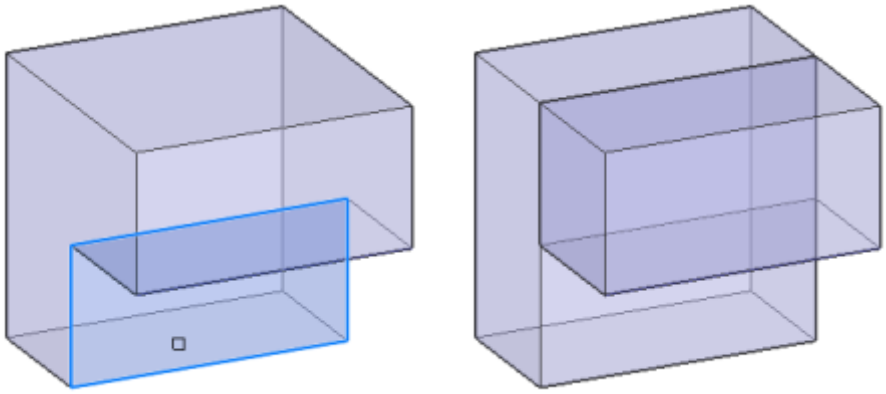
The section plane is perpendicular to the XY-plane of the World coordinate system (WCS).

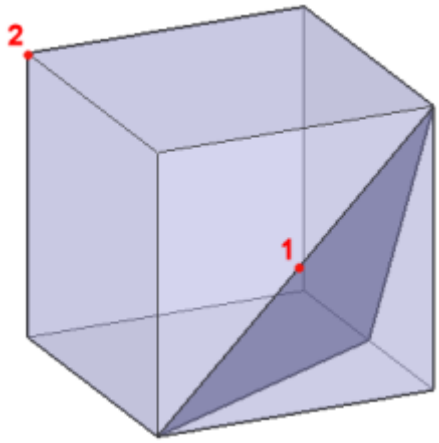
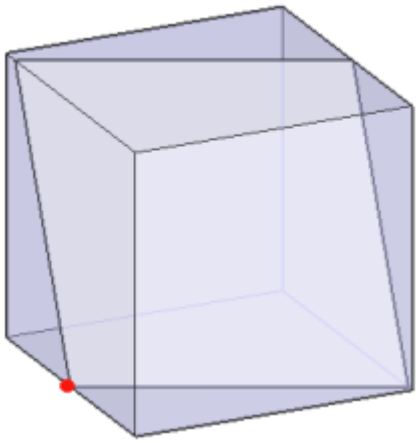
The sliced solid can be separated and its two parts manipulated as independent entities:

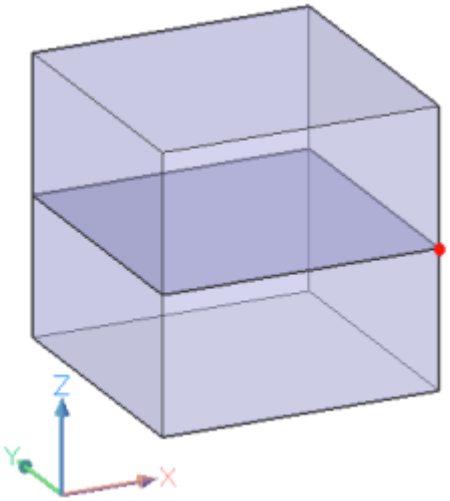
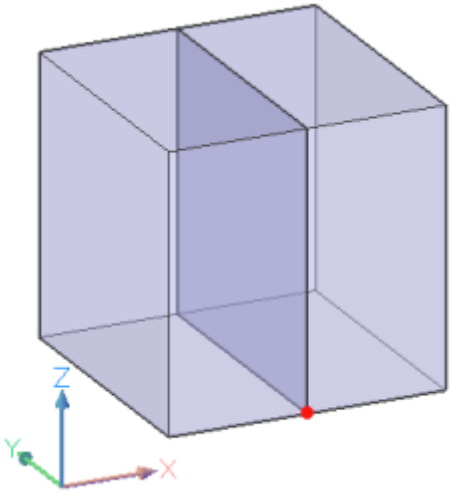


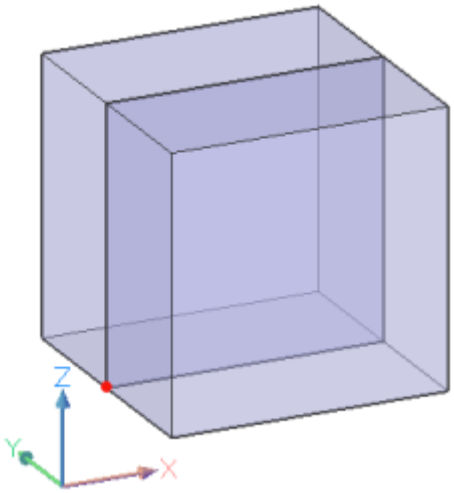
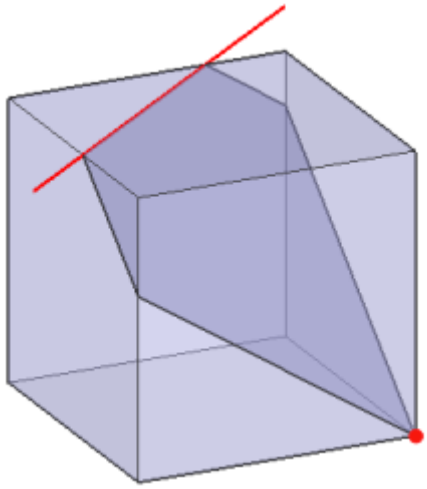
Command Options

Option	Description
Select entities	Specifies the entities that will be sliced. Choose one or more 3D solids; this command does not work with 2D regions.
3points	Specifies the first of three points that define the slicing plane. Pick a point or enter x,y,z coordinates; prompts you: Specify first point on plane - pick a point. Specify second point on plane - pick another point. Specify third point on plane - pick a third point. 
Object	Slices the 3D model with an entity that intersects the 3D solid; prompts you: Select a circle, ellipse, arc, 2D-spline, 2D-polyline, planar 3d Solid face, planar Surface face, or planar region:- choose an entity. You can also use elliptical arcs. The entity defines the plane that slices the 3D solid.

	 Using a planar 3d solid face: Make sure the *Select Faces* option of the SELECTIONMODES system variable is checked. 
Surface	Slices the 3D solid using a surface; prompts you: Select a surface - select a surface entity.
Zaxis	Slices the 3D model with a plane defined by one point on the plane, and a second on the plane's normal (z axis); prompts you: Specify a point on the section plane - pick a point (1) to define the location of the section plane. Specify a point on the Z-axis (normal) of the plane - pick another point (2) to define the normal. The section plane is perpendicular to the line trough the specified points.

	 Section plane defined by a point: centroid of the front face of the cube (1) and the Z-Axis through this point and a corner of the cube (2). The section plane is perpendicular to the line 1-2.
View	Slices the 3D model with a plane defined by the current 3D viewpoint; prompts you: Specify a point on the current view plane - pick a point to define the height of the plane.  The section plane is perpendicular to the view direction and passes through the picked point (1).
XY	Slices the 3D model with a plane parallel to the XY-plane of the current coordinate system; prompts you: Specify a point on the XY-plane - pick a point to define the position of the slicing plane.

	
YZ	Slices the 3D model with a plane parallel to the YZ-plane of the current coordinate system; prompts you: Specify a point on the YZ-plane - pick a point to define the position of the slicing plane. 
ZX	Slices the 3D model with a plane parallel to the ZX-plane of the current coordinate system; prompts you: Specify a point on the ZX-plane - pick a point to define the position of the slicing plane.

	
Line-point	Slices the 3D model with a plane defined by a linear entity and a point; prompts you: Select a line, 2d-polyline segment, linear edge of 3dSolid or surface, or axial entity to use its axis - select a linear entity. Specify point on slicing plane - pick a point. 
Specify a point on desired side of the plane or Both	Specifies which of the two portions to keep. Choose one of the parts; the other one is erased. Or type B to retain both parts.

Related Commands

Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Region - creates 2D regions from 2D entities.
Section - creates 2D regions from 3D models.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.

Snap

Sets snap properties; toggles snap mode.

Accessing the Command

command bar: *snap*
transparent: *'snap*
shortcut: *F9*
alias: *sn*
menu bar: *Settings | Snap*
toolbar: *Settings |*
status bar: *SNAP*

: ***snap***
Prompts you in the command bar:

Snap is off (x and y = 10): Set snap spacing or [snap ON/Rotate/snap Style/Aspect]: (Enter a number for spacing, or enter an option.)

Snap acts as a cursor resolution, constraining cursor movement to discrete distances, such as 0.5".

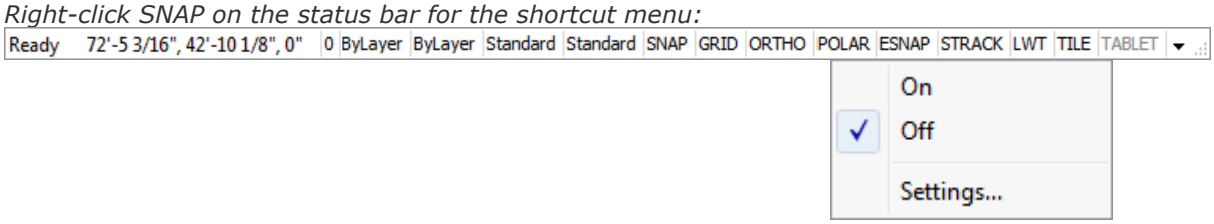
Command Options

Option	Description
Set snap spacing	Specifies the snap spacing: • Enter a single number to specify the spacing between snap points. • Enter x,y coordinates to show the snap spacing; prompts you: Second point - pick another point or enter x,y coordinates.
ON	Turns on snap.
OFF	Turns off snap.
Rotate	Rotates the snap grid; prompts you: Base point for snap grid - specify the point about which the snap grid rotates. Pick a point or enter x,y coordinates Rotation angle - specify the angle to which to rotate the snap grid. This option also rotates the grid, crosshair cursor, and ortho axes.
Style	Toggles between standard and isometric style. The isometric style affects the grid, crosshair cursor, and adds the Isocircle option to the **Ellipse** command.
Isometric/<Standard>	Type I or S: • Isometric - switches to isometric snap grid. • Standard - switches to rectangular grid.

Aspect/Vertical spacing	Type A or V: • Aspect - specifies spacing independently in the x and y directions; prompts you: Horizontal spacing - specify the space between snap points in the x direction. Vertical spacing - specify the space between snap points in the y direction. • Vertical spacing - specifies the same spacing for the x and y directions.
Aspect	Specifies spacing independently in the x and y directions; prompts you: Horizontal spacing - specify the space between snap points in the x direction. Vertical spacing - specify the space between snap points in the y direction.

Status Bar Menu

Right-click SNAP on the status bar for the shortcut menu:



Shortcut Menu	Description
On	Turns on snap.
Off	Turns off snap.
Settings	Opens the Snap/Grid section of the Settings dialog box. Drawing Drafting Drawing units Coordinate input Snap/Grid Snap angle: 0.0000 Snap base: 0", 0" X: 0" Y: 0" Snap isometric pair: Left Snap mode: Snap on (for current viewport) Snap style: Rectangular snap Snap type: Grid snap Snap unit: 1/2", 1/2" X: 1/2" Y: 1/2"

Snap angle	Sets the angle of the snap in the current UCS. Also affects the angle of the grid display and ortho mode.
Snap base	Specifies the x,y-coordinates about which the snap rotates. Also affects the angle of the grid display and ortho mode.
Snap isometric pair	Specifies the isometric plane; click the droplist and choose one: • Left - left isometric plane. • Top - top isometric plane. • Right - right isometric plane. This setting requires Snap Style to be set to Isometric Snap.
Snap mode	Toggles snap mode: • On - turns on snap. • Off - turns off snap.
Snap style	Specifies the snap style; click the droplist and choose one: • Rectangular snap - regular snap mode. • Isometric snap - isometric mode.
Snap type	Specifies the snap type; click the droplist and choose one: • Grid snap - uses distances along the x and y axes. • Polar snap - uses relative distance and angle.
Snap unit	Specifies the snap increment distance: • X - spacing in the x direction. • Y - spacing in the y direction. There is no snap in the z direction.

Related Commands

Settings - sets parameters related to snap mode.
Grid - controls the display of the grid.
Isoplane - switches to the next isometric plane.

Solid

Draws 3 and 4-sided regular and irregular 2D polygons; does not draw 3D solids.

Accessing the Command

command bar: *solid*
aliases: *plane, so*
menu bar: *Draw | Solid*
toolbar: *Draw | Polygons |*

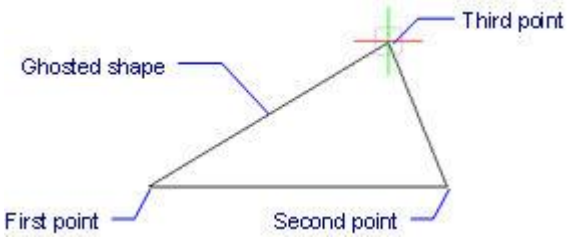
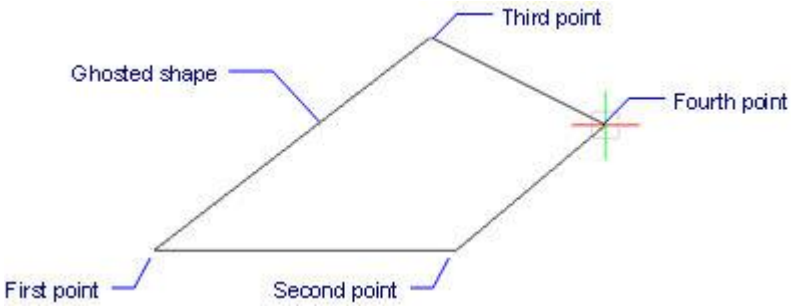
: ***solid***
Prompts you in the command bar:
Set first point of plane or [Rectangle/Square/Triangle]: (Pick a point, or enter an option.)

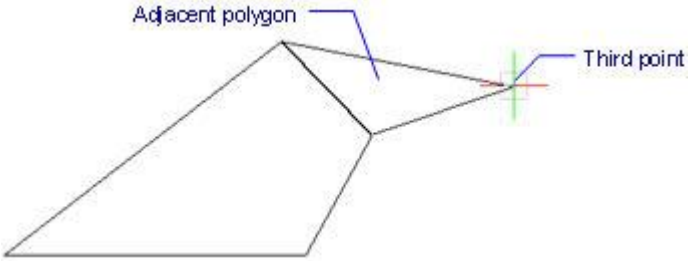
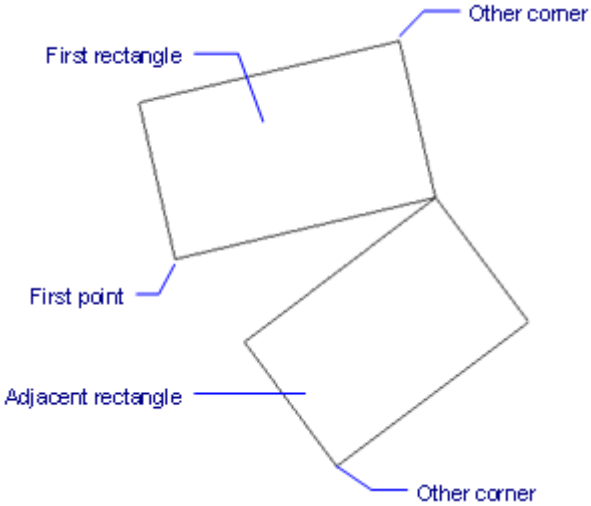
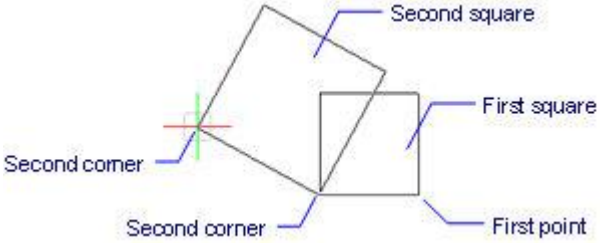
Second point: (Pick another point.)

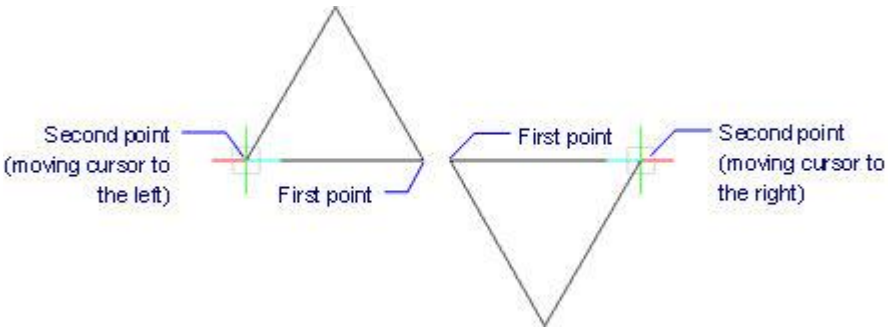
Third point of plane: (Pick another point.)

Fourth point: (Pick another point to draw a quadrangle, or press Enter to draw a triangle and exit the command.)
Third point of plane: (Pick another point to draw an adjoining triangle, or press Enter to exit the command.)

Command Options

Option	Description
Set first point of plane	Specifies the first corner of the polygon. Pick a point or enter x,y coordinates.
Second point	Specifies the second corner of the polygon. BricsCAD ghosts the shape and size of the polygon.
Third point of plane	Specifies the third corner of the polygon. 
Fourth point	Specifies the fourth corner of the polygon.  To draw a triangle, press Enter at the prompt.

Third point on plane	Starts drawing an adjacent polygon. The third and fourth points of the first polygon become the first and second points of the second polygon.  To end the command, press Enter at the this prompt.
Rectangle	Draws one or more connected rectangles; prompts you: First point of plane - Pick a point. Other corner of rectangle - Pick a point. Rotation angle for plane - Pick a point or enter an angle. Other corner of rectangle - Pick a point or press Enter to stop. 
Square	Draws one or more connected squares; prompts you: First point of plane - Pick a point. Second corner of square - Pick a point. Second corner of square - Pick a point or press Enter to stop. 
Triangle	Draws one or more connected equilateral triangles; prompts you: First point of plane - Pick a point. Second corner of triangle - Pick a point. Cursor movement determines the orientation of the triangle.

	Move cursor to the: • Right - triangle is drawn upside down. • Left - triangle is drawn rightside up.  Second corner of triangle - Pick a point or press Enter to stop.

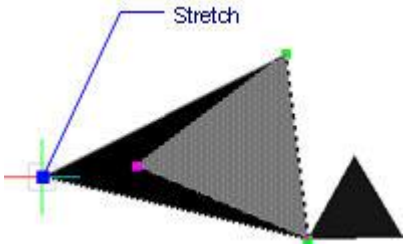
Procedures

Drawing Solids

Grips Editing

2D solids can be edited directly through grips:

1. Select the solid. Notice that it has one grip at each vertex.
2. Drag a grip to stretch the solid.



Related Commands

Fill - toggles the display of solid areas.
Trace - draws wide lines.
BHatch and Hatch - place solid-filled hatch patterns through a dialog box.
-BHatch and -Hatch - place solid-filled hatch patterns through the command bar.
Boundary and -Boundary place a polyline boundary inside closed areas.
Properties - changes the properties of solid.

SolidEdit

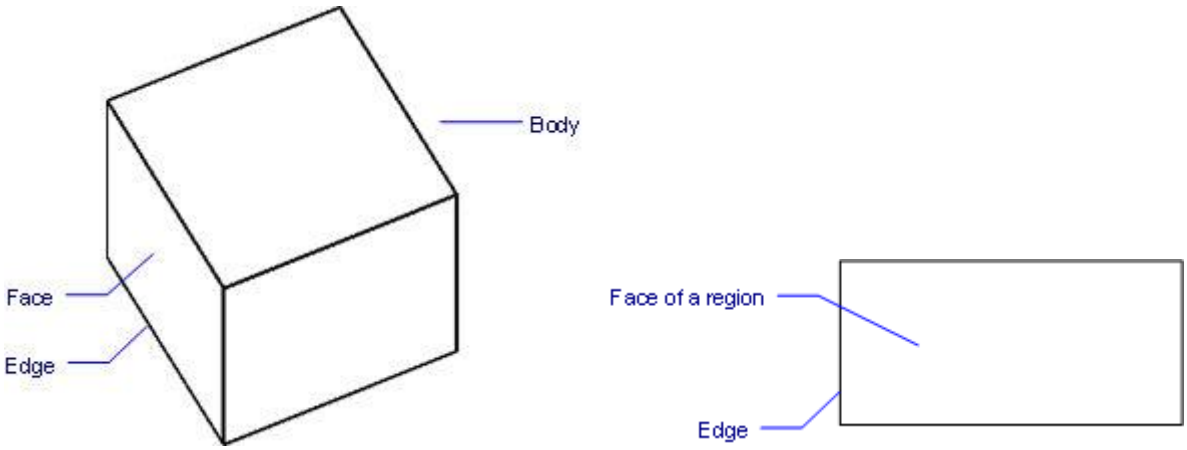
Edits 3D solids and 2D regions.

Accessing the Command

command bar: *solidedit*
menu bar: *Model | 3D Solid Editing*
toolbar: *3D Solid Editing |*

: ***solidedit***
Prompts you in the command bar:

Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: (Enter an option, or press Enter to end the command.)



Face editing options:

Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: (Enter an option, or press Enter to return to the previous prompt.)

See Face Editing Options section below.

Edge editing options:

Enter an edge editing option: Copy/coLor/Undo/<eXit>: (Enter an option, or press Enter to return to the previous prompt.)

See Edge Editing Options section below.

Body editing options:

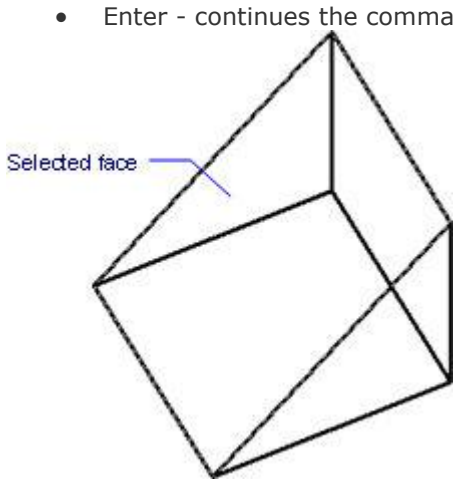
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>: (Enter an option, or press Enter to return to the previous prompt.)

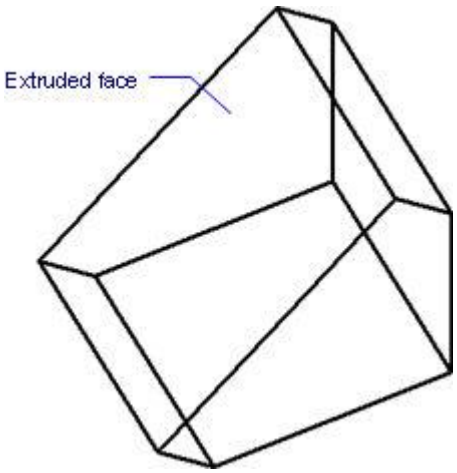
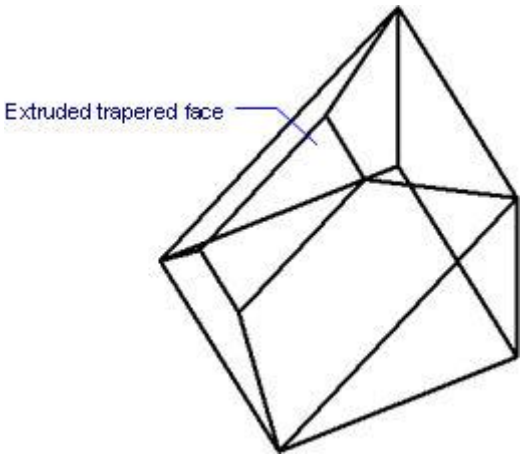
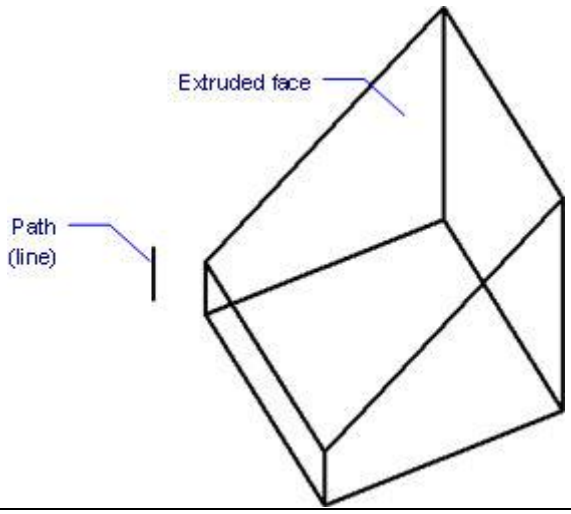
See Body Editing Options section below.

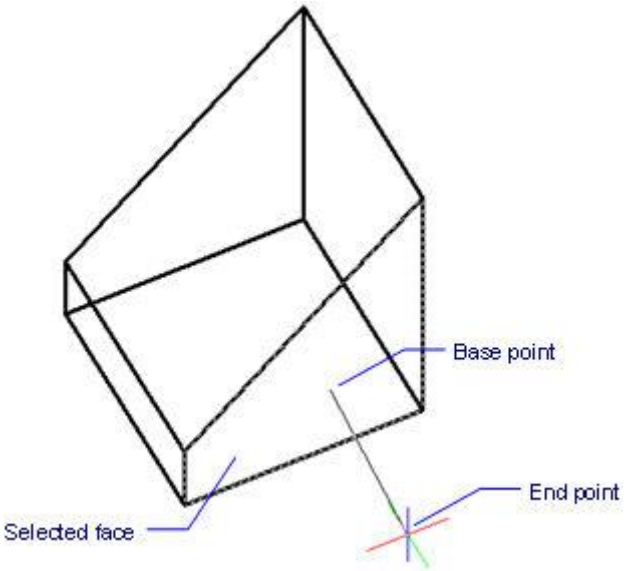
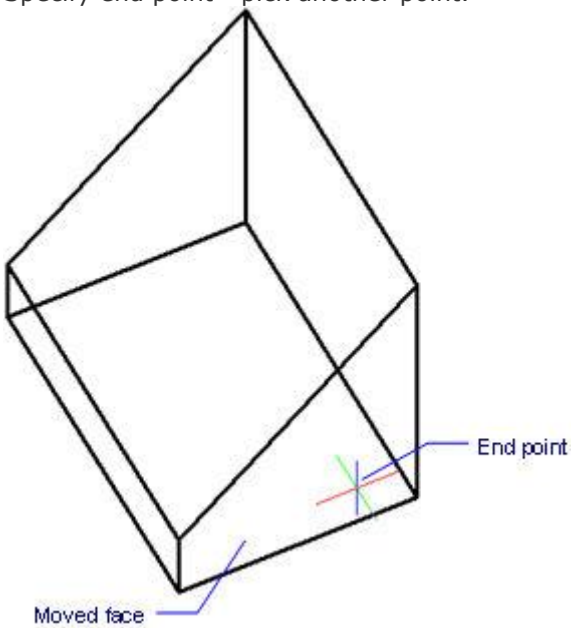
Command Options

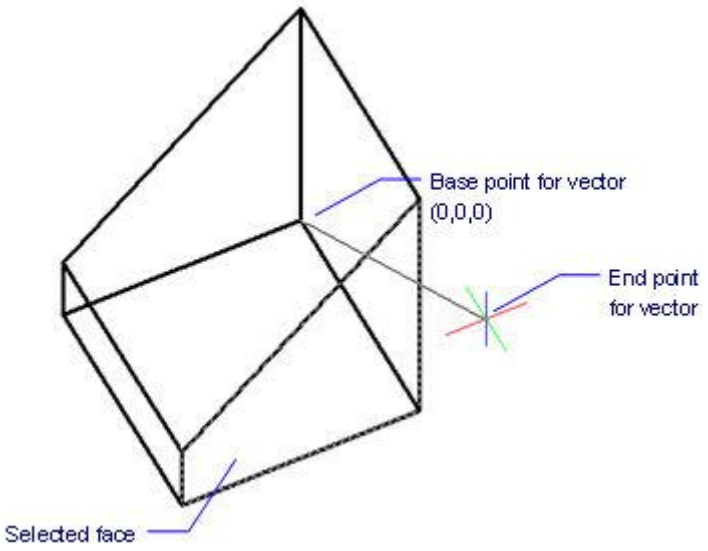
Option	Description
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>:	Specifies the type of editing to perform. Enter an option.
Face	Edits faces of 3D solids and 2D regions; prompts you:

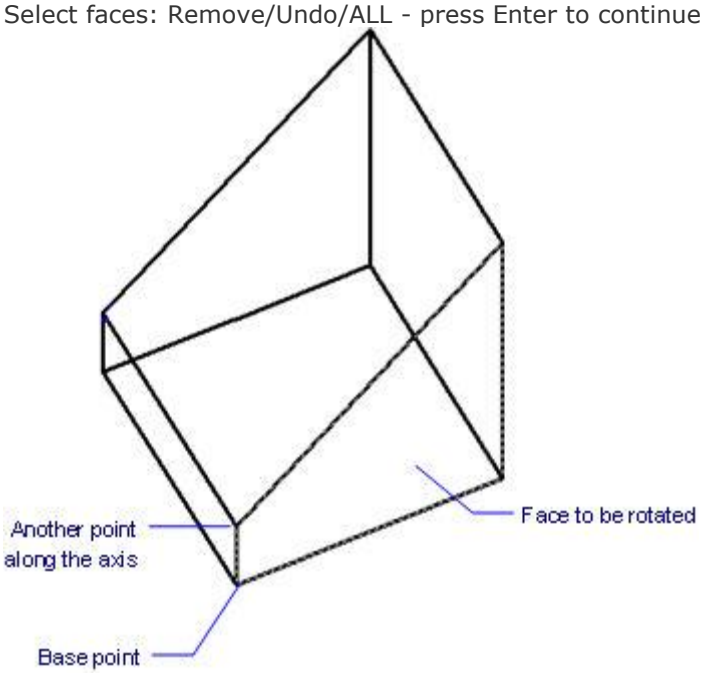
	Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit> - enter an option. Press Enter to return to the previous prompt. See Face Editing Options section below.
Edge	Edits edges of 3D solids and 2D regions; prompts you: Enter an edge editing option: **Copy**/**coLor**/**Undo**/<**eXit**> - enter an option. Press Enter to return to the previous prompt. See Edge Editing Options section below.
Body	Edits 3D solids (a.k.a. bodies); prompts you: Enter a body editing option **Imprint**/**seParate**/**Shell**/**cLean**/**Check**/**Undo**/<**eXit**> - enter an option. Press Enter to return to the previous prompt. See Body Editing Options section below.
Undo	Undoes the last editing operation.
eXit	Exits the command.

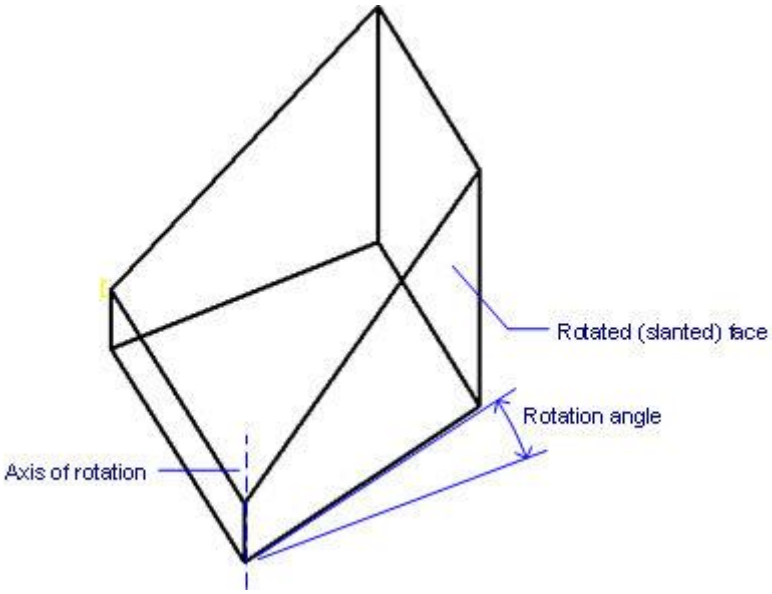
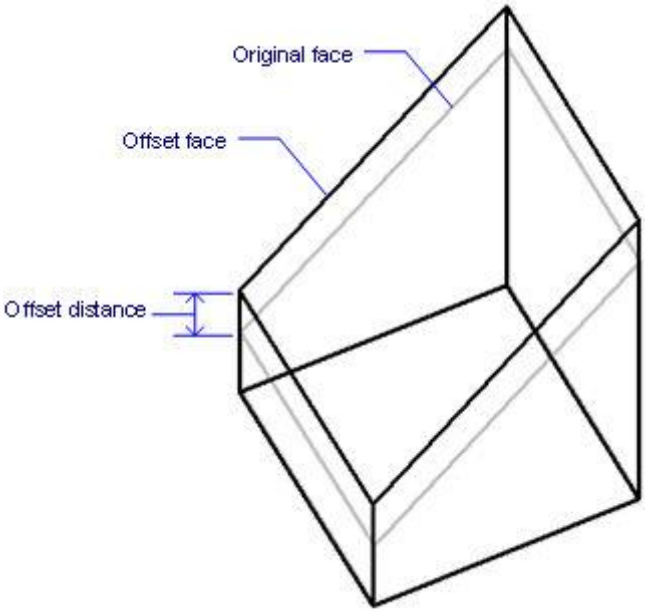
Face Editing Options **Enter a face editing option:** Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/**<**eXit**>:**	
Extrude	Extrudes the selected face or region; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change: • Pick one or more faces. • Remove - removes faces from the selection set. • Undo - undoes the last selection operation. • ALL - selects all faces in the drawing. • Enter - continues the command to the next prompt:  Specify height of extrusion or **Path** - specify a distance, or enter P to select an entity that defines the path: • Positive distance - extrudes outwards from the face (creates a post). • Negative distance - extrudes into the body (creates a hole).

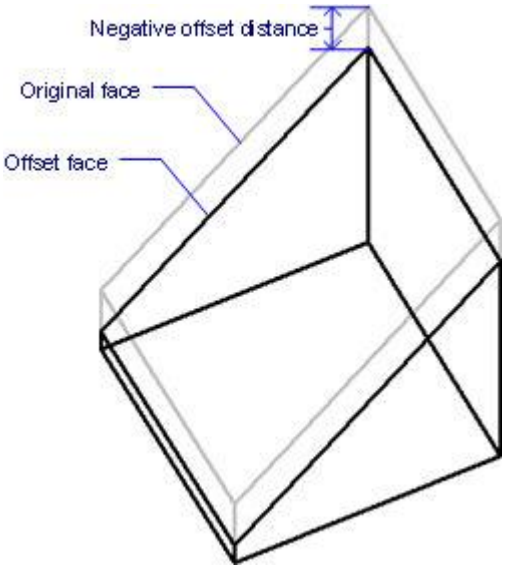
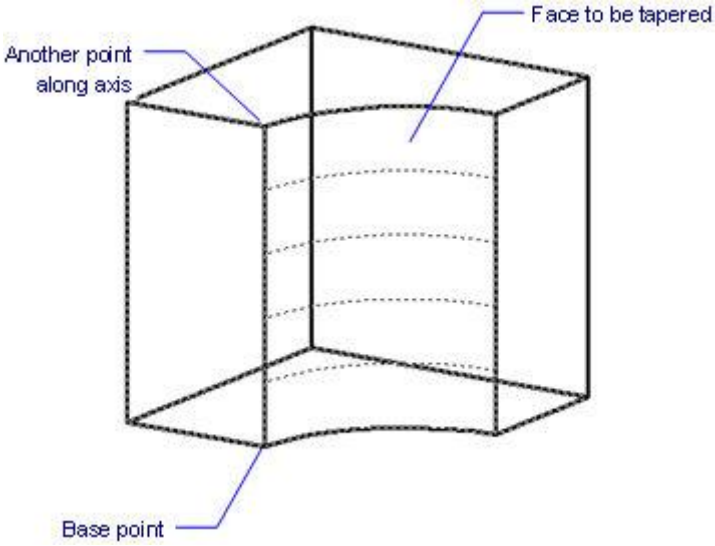
	 Specify angle of taper for extrusion - specify an angle, or enter 0 for no taper: • Positive angle - tapers inwards (gets narrower). • Negative angle - tapers outwards (gets wider).  See Extrude command.
Path	Specifies the entity for defining the extrusion shape and direction; prompts you: Select extrusion path - choose a open entity, such as line, arc, polyline, or spline. 

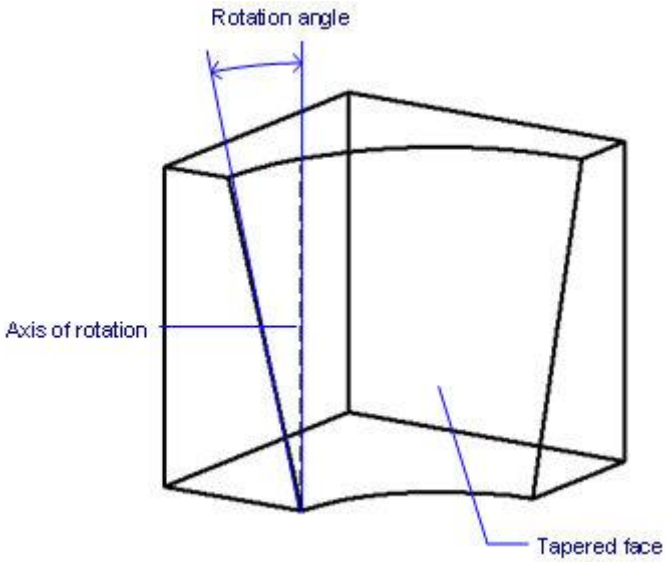
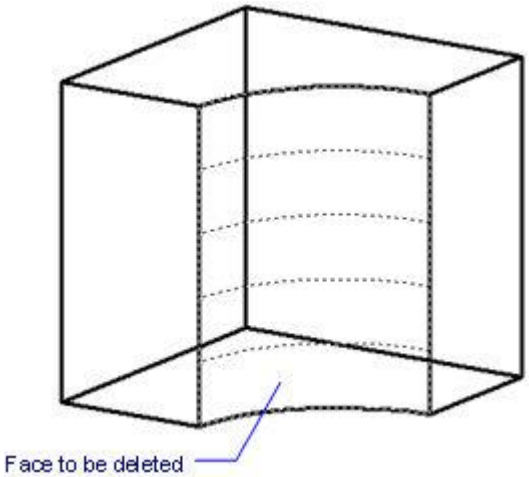
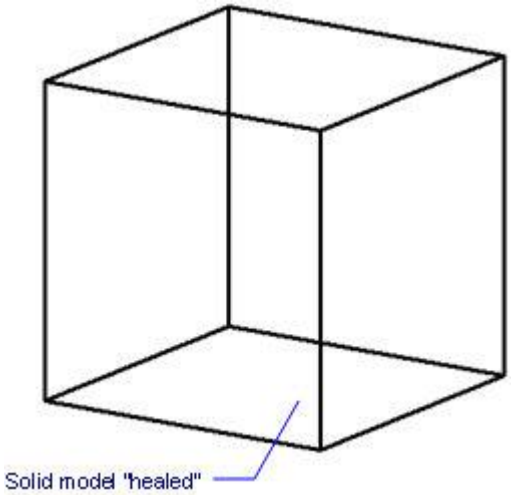
Move	Moves the selected face; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.  Specify <Base point>/**Vector** - pick a point, enter x,y,z coordinates, or type V. Specify end point - pick another point.  See Move command.
Vector	Specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point; prompts you: Specify end point - pick a point or enter x,y,z coordinates.

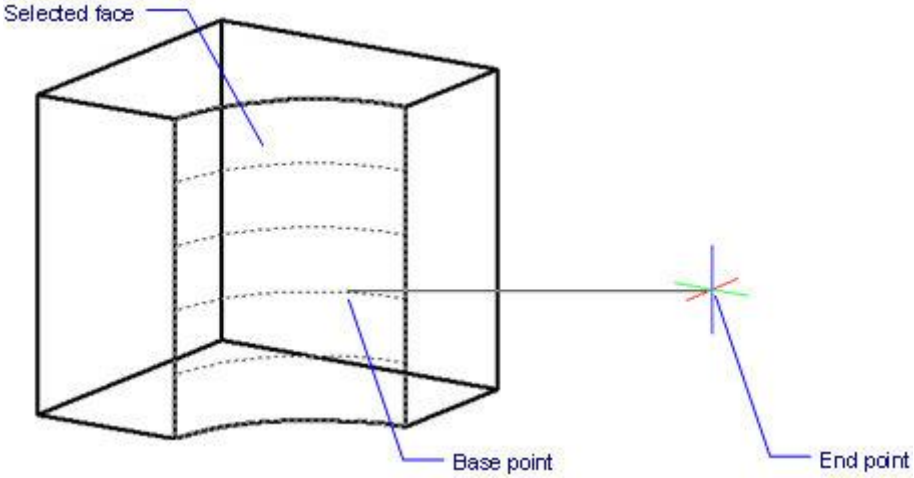
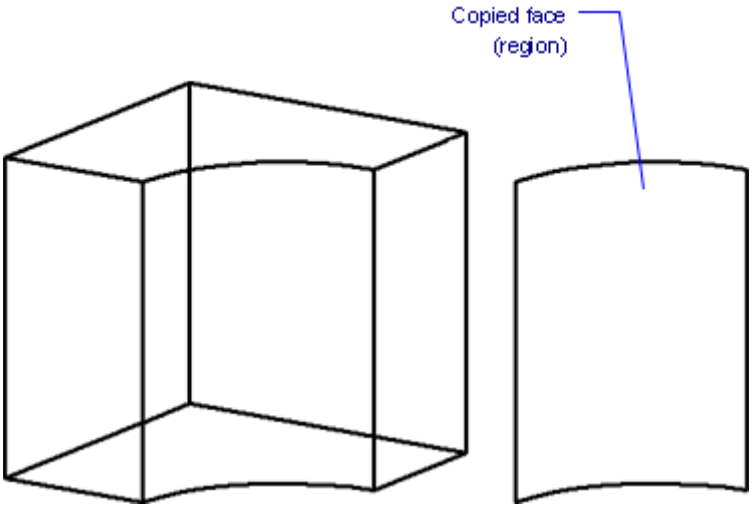


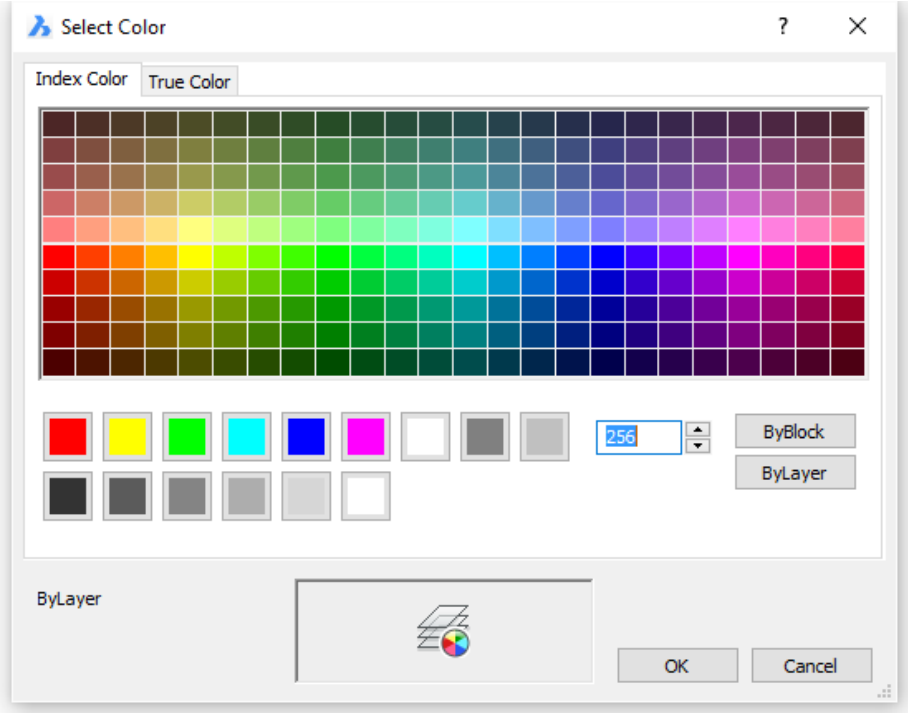
Rotate	Slants the selected face about an axis; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.  Specify the base point - pick the first point of the axis of rotation. Specify another point along the axis - pick the axis of rotation's second point. Specify the rotation angle - enter an angle: • Positive angle - face rotates counterclockwise around the axis. • Negative angle - face rotates clockwise.

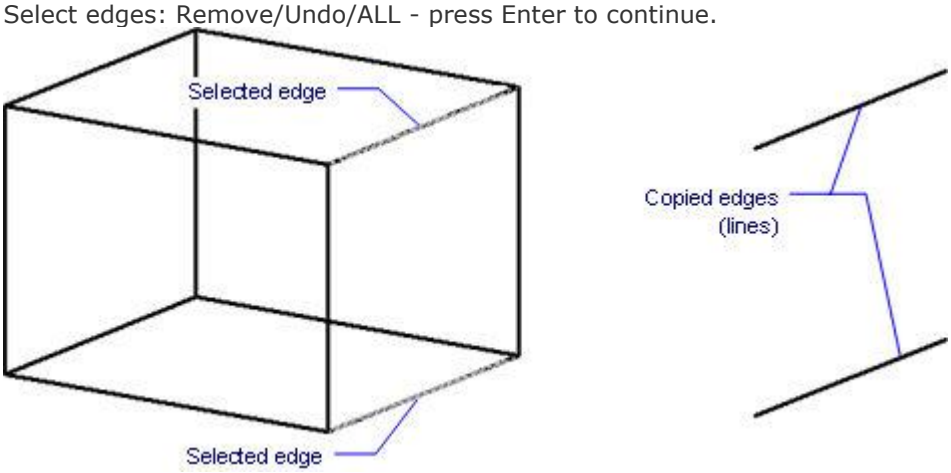
	 See Rotate command.
Offset	Extends the selected face; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.  Specify offset distance - enter a distance: • Positive distance - face is offset outwards (3D body becomes larger). • Negative distance - face is offset inwards (3D body becomes smaller).

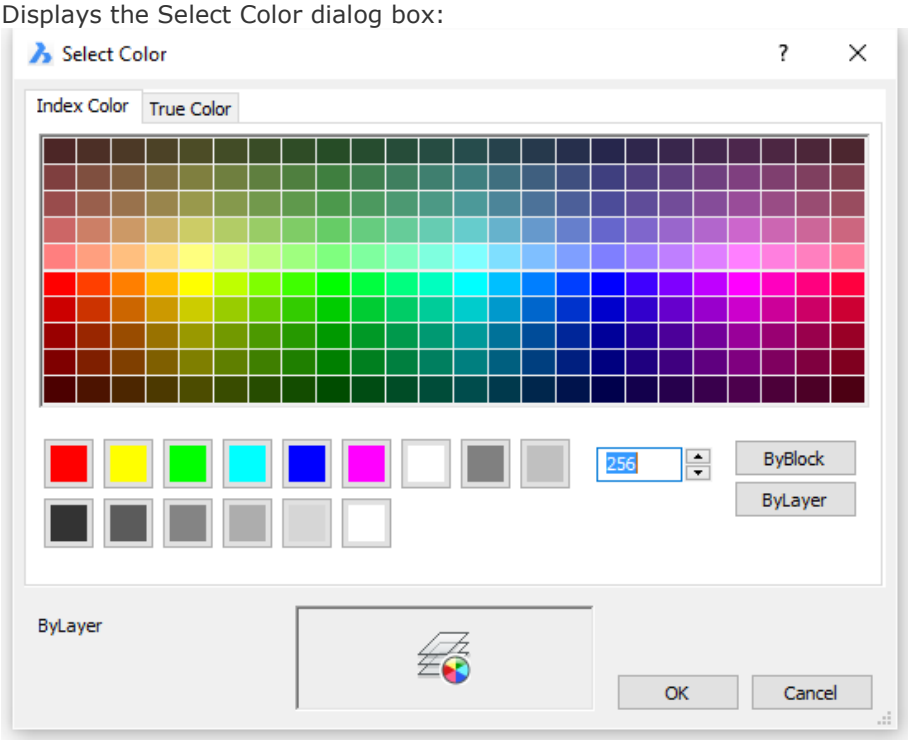
	 See Offset command.
Taper	Tapers the selected face; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.  Specify the base point - pick the first point of the axis of rotation. Specify another point along the axis - pick the axis of rotation's second point. Specify the rotation angle - enter an angle: • Positive angle - face rotates counterclockwise around the axis. • Negative angle - face rotates clockwise.

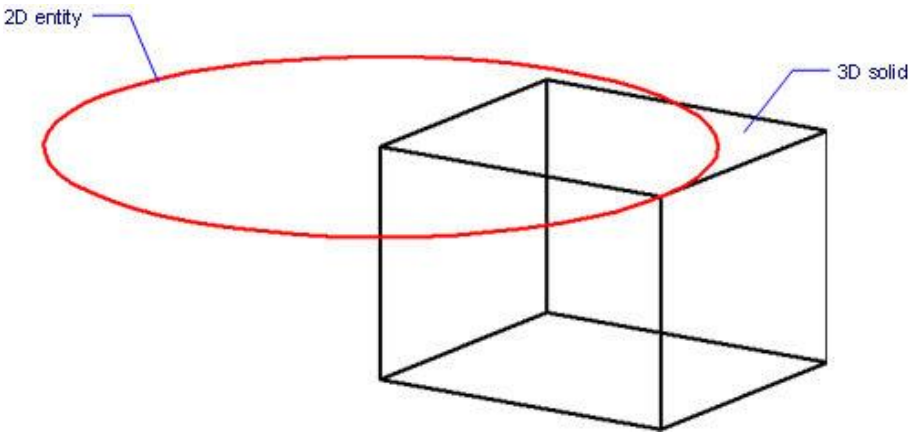
	
Delete	Deletes the selected face, and then repairs the model; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change.  Select faces: Remove/Undo/ALL - press Enter to continue.  The selected faces are deleted, and BricsCAD repairs the 3D solid to keep it solid.

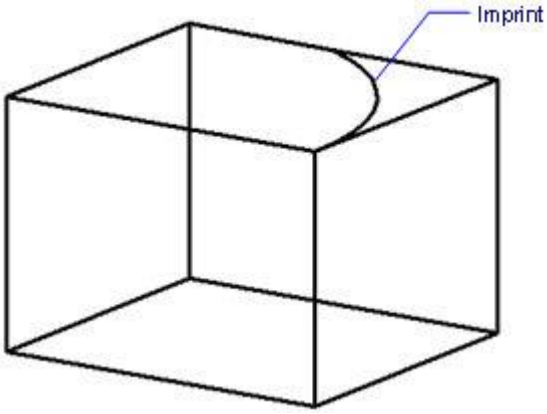
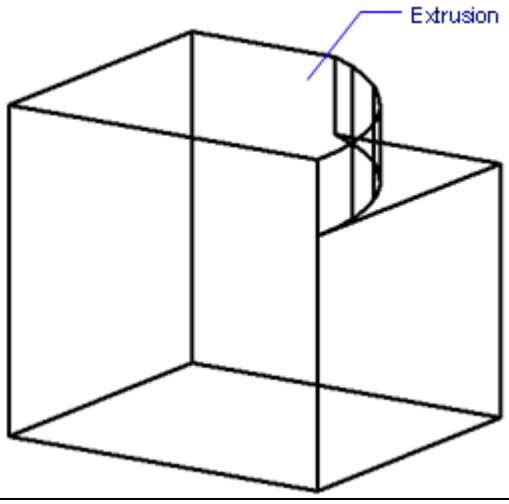
	This option cannot remove inate faces, such as those on the cube illustrated above.
Copy	Copies faces; selected faces turn into regions. Prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.  Specify <Base point>/**Vector** - pick a point, enter x,yz, coordinates, or type V. Specify end point - pick another point.  To copy edges to lines, use the Edge |Copy option.
Vector	Specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point; prompts you: Specify end point - pick a point or enter x,y,z coordinates.
coLor	Changes the color of the selected face; prompts you: Select faces: Remove/Undo/ALL - choose the face(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue. Displays the Select Color dialog box:

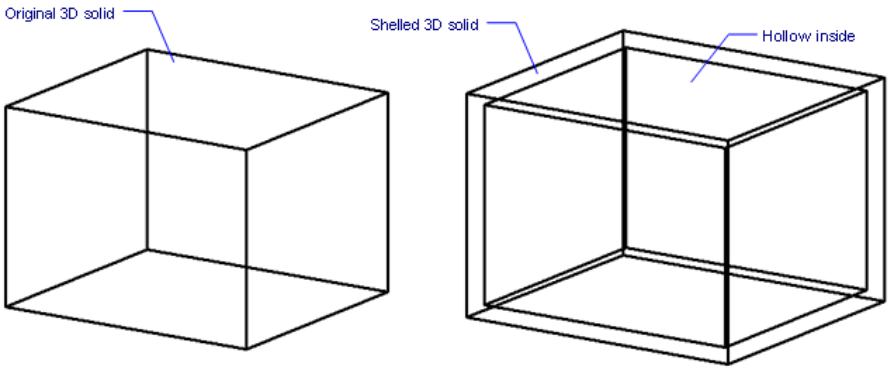
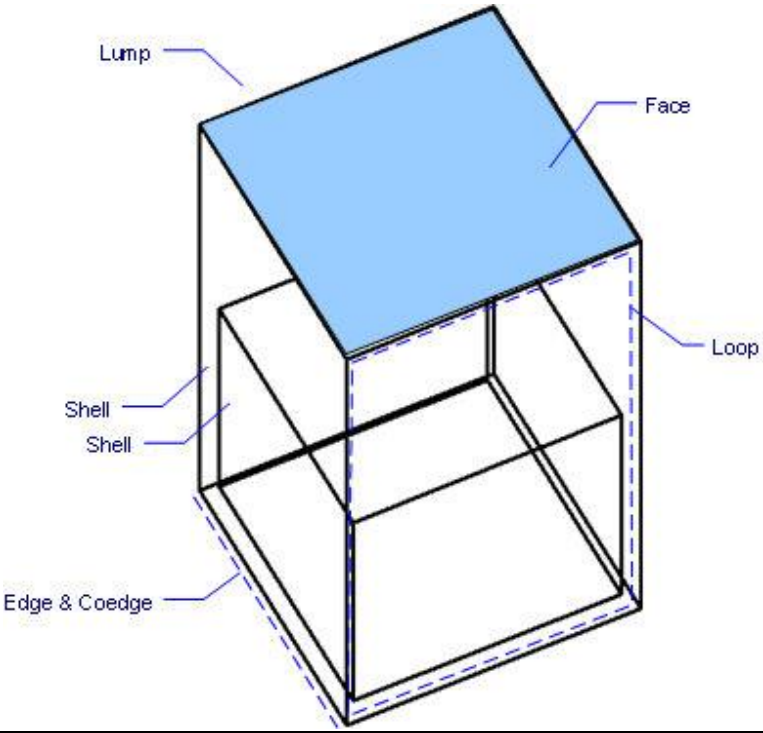
	 Choose a color, and then click OK. (Note: this option does not work.)
Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Edge Editing Options **Enter an edge editing option:** Copy/coLor/Undo/<eXit>:	
Copy	Copies the selected edge, and turns it into a line; prompts you: Select edges: Remove/Undo/ALL - choose the face(s) to change. Select edges: Remove/Undo/ALL - press Enter to continue.  Specify <Base point>/Vector - pick a point, enter x,y,z coordinates, or type V. Specify end point - pick another point, or enter x,y,z coordinates. Copied edges are turned into line entities.
coLor	Changes the color of the selected edge; prompts you: Select faces: Remove/Undo/ALL - choose the edge(s) to change. Select faces: Remove/Undo/ALL - press Enter to continue.

	Displays the Select Color dialog box:  Choose a color, and then click OK. (Note: this option does not work.)
Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Body Editing Options **Enter a body editing option** Imprint/seParate/Shell/cLean/Check/Undo/<eXit>:	
Imprint	Imprints 2D entities onto 3D solids; prompts you: Select 3d solid - choose a 3D solid. Select an object to imprint- choose a 2D entity lying on or intersecting at least one face.  Delete the source object Yes/<No> - type Y or N. • Yes - deletes the 2D entity. • No - does not delete the 2D entity, which can be reused.

	Select an object to imprint - choose another 2D entity, or press Enter to end.  The 2D entity must lie on, or intersect, one of the 3D solid's faces. You can extrude the imprint. 
seParate	Separates disjointed 3D solids; prompts you: Select 3d solid - choose a 3D solid. This option does not work on bodies created through Boolean operations (Union, Intersect, Subtract).
Shell	Turns the 3D solid into a hollow entity (shelled); prompts you: Select 3d solid - choose a 3D solid. Remove faces: Add/Undo/ALL - press Enter; removing faces creates unexpected results. Enter the shell offset distance - specify a distance: • Positive distance - shells body outwards. • Negative distance - shells body inwards.

	 If the shell offset distance is larger than the 3D solid, this command fails.
cLean	Cleans the 3D solid; prompts you: Select 3d solid - choose a 3D solid. The cleaning operation checks for errors in the 3D solid.
Check	Reports on the 3D solid; prompts you: Select 3d solid - choose a 3D solid. A typical report looks like this: **checked:** **1 lumps** **2 shells** **0 wires** **12 faces** **12 loops** **48 coedges** **24 edges** **16 vertices** 

Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Related Commands

BimSplit - splits segments of wall-like solids into separated solids.
Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Imprint - mprints 2D entities onto planar faces of 3D solids and surfaces.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Revolve - revolves 2D entities into 3D solid models.
Section - creates 2D regions from 3D models.
Slice - cuts 3D solids into two parts.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

Solprof

Creates a hidden line representation of 3D solids in a layout viewport.
This command can only be used in a layout tab.

Accessing the Command

Select a layout, then double click inside a layout viewport (Model Space, with floating viewports).
command bar: *solprof*
menu bar: *Model* | *Solprof*
toolbar: *Sections* |

: ***solprof***
Prompts you in the command bar:

Select Entities: select 3D solids. Press Enter to stop select entities.

Use separate layer for hidden lines? [Yes/No] <Yes>: choose an option.

Project results onto plane? [Yes/No] <Yes>: choose an option.

Hide tangential edges? [Yes/No] <Yes>: choose an option.

The selected 3D solids are projected onto a plane parallel with the layout viewport.

Two blocks are created one for the visible, one for the hidden lines.

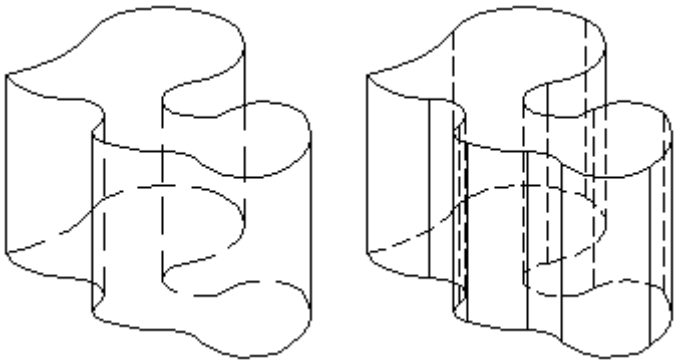
The blocks are placed on two newly created layers: one for the visible lines and one for the hidden lines.

- The layer containing the visible lines is named: PV-<viewport handle>.
- The layer for the hidden lines is named: PH-<viewport handle>.

Select the viewport, then run the List command to see the handle of a viewport.

Command Options

Option	Description
Use separate layer for hidden lines?	Yes: Two blocks are created: one for the visible lines, one for the hidden lines. The block of the visible lines is created on the PV-<viewport handle> layer. The block of the hidden lines is created on the PH-<viewport handle>. No: All profile lines of each selected solid are treated as visible lines, even if some of the profile lines are hidden behind another solid. Only one block is created on the PV-<viewport handle> layer.
Project results onto plane?	Yes: The profile lines are projected onto the view plane (= the plane perpendicular to the view direction and passing through the origin of the current UCS). No: No projection. Each profile line remains at its 3D location.
Hide tangential edges?	A tangential edge is the transition line between two tangent faces. It is an imaginary line between the two faces. In most cases tangential edges need to be removed.

	 *Tangential edges removed (left) - tangential edges displayed (right).* Yes: Tangential edges are removed. No: Tangential edges display.

Related Commands

Flatshot - creates a hidden line representation of 3D solids in model space.
Hide - removes hidden lines from 3D entities.
Shademode - sets the shading style for 3D drawings.
ViewExport - exports the content of drafting views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.

Spell

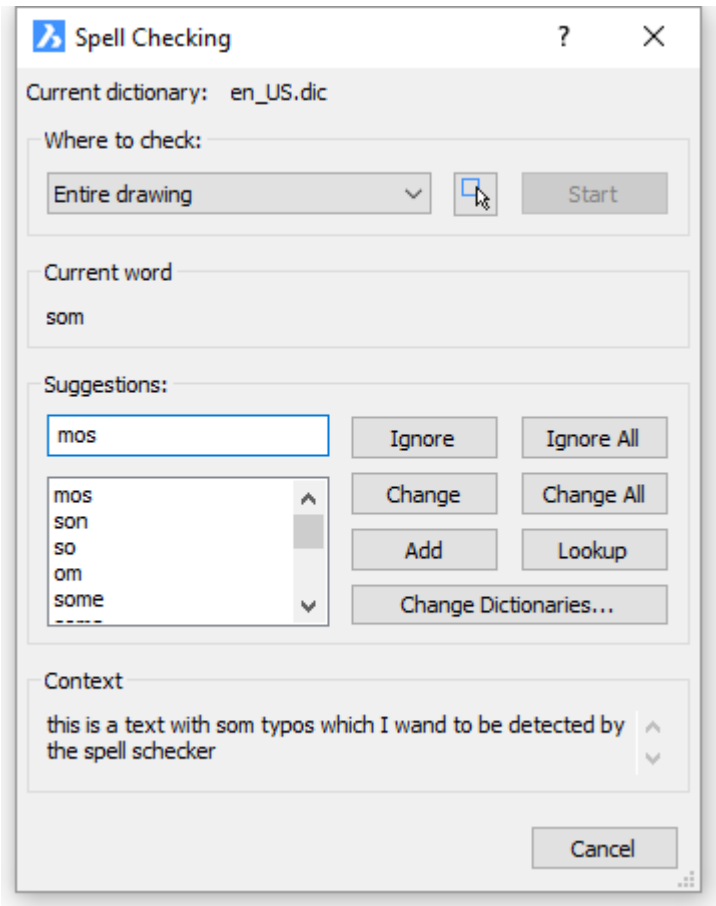
Checks the spelling of words in drawings.

Accessing the Command

command bar: *spell*
alias: *sp*
menu bar: *Tools | Check Spelling*

: ***spell***

Displays a dialog box:

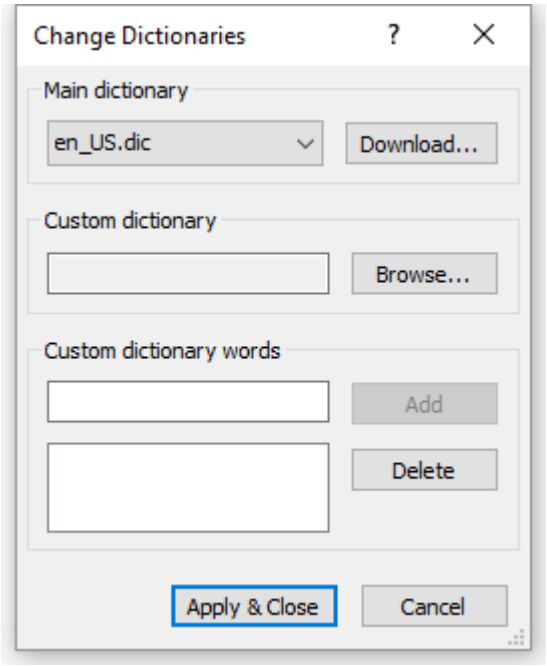


Click Start.

Command Options

Option	Description
Where to check	Checks text in: • Entire drawing - checks all text in the drawing except text on frozen layers. • Selected entities - checks selected text; click the Select Entities button to choose text.

Select entities	Selects the text to be checked; prompts you: Select entities - choose one or more pieces of text; press Enter to return to the dialog box.
Start	Starts spell checking.
Suggestions	Lists the suggested replacements for misspelled words. Choose one, and then click Change or Change All.
Ignore	Ignores the spelling of the word.
Ignore All	Ignores all instances of the word.
Change	Changes the word to the one selected from the Suggestions list.
Change All	Changes all instances of the word to the one selected from the Suggestions list.
Add	Adds the word to the word list.
Lookup	Provides additional alternative spellings for the word selected in the Suggestions list.
Change Dictionaries	Changes the dictionary used for spell checking; displays the Change Dictionaries dialog box.



Change Dictionaries	Description
Main Dictionary	Specifies the dictionary to use for spell checking. To change dictionaries: 1. Click the droplist. 2. Choose another dictionary.
Download	Accesses additional dictionaries from the OpenOffice.org Wiki. Opens the Web browser at https://extensions.openoffice.org/dictionaries
Custom dictionary	Specifies the name of a custom dictionary. *Custom dictionaries* are used for discipline-specific words, such as medical or mechanical.

Browse	Selects a CUS custom dictionary file. Opens the Select Custom Dictionary dialog box: 1. Choose a CUS file. 2. Click Open.
Custom dictionary words	Enters words to be stored in the custom dictionary.
Add	Adds the word to the custom dictionary. Words are also added when Add is clicked on the Spell Checking dialog box.
Delete	Removes the word from the custom dictionary.

Procedures

To check the spelling
To change dictionaries
To add a new main dictionary
To add a new custom dictionary

Related Commands

Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Find - finds and replaces text.
Properties - modifies the properties of text.

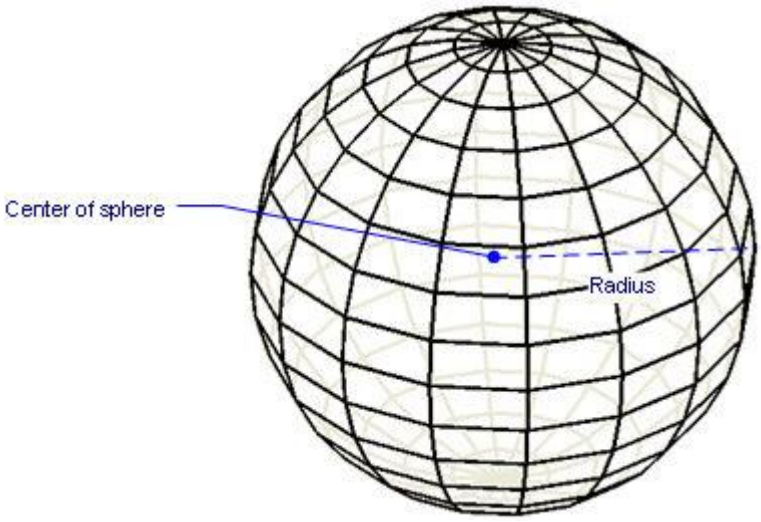
Sphere

Creates a sphere as a polygon mesh (Classic); creates a sphere as a 3D solid (Pro and Platinum).

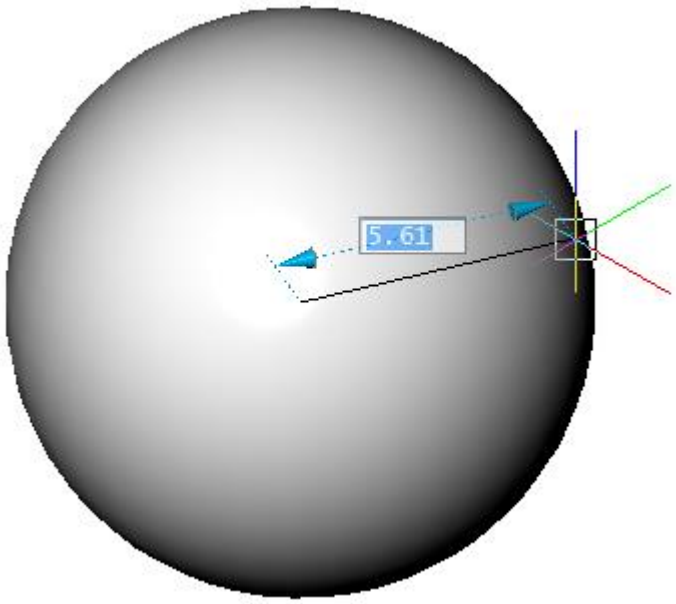
Accessing the Command

command bar: *sphere*
menu bar: *Model | 3D Solids | Sphere*
toolbar: *3D Solids |*

: ***sphere***
Prompts you in the command bar:
Center of sphere: (Pick a point, or enter coordinates.)
Set radius of sphere or [Diameter]: (Specify a distance, or type D.)



If Dynamic Dimensions are active, you can enter the radius in the dynamic dimension entry field.



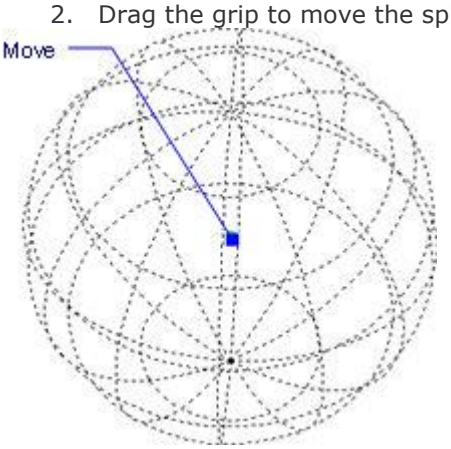
Command Options

Option	Description
Center of sphere	Specifies the center point of the sphere; enter x,y- or x,y,z-coordinates, or pick a point in the drawing.
Radius of sphere	Specifies the radius.
Diameter	Specifies the diameter.

Grips Editing

Spheres can be edited directly through grips:

1. Select the sphere. Notice that it has one grip.
2. Drag the grip to move the sphere.



Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

Spline

Draws open and closed splines.

Accessing the Command

command bar: *spline*
alias: *spl*
menu bar: *Draw | Spline*
toolbar: *Draw |*

: ***spline***

Prompts you in the command bar:

First point for spline: (Pick a point, or enter x,y coordinates.)

Second point: (Pick another point.)

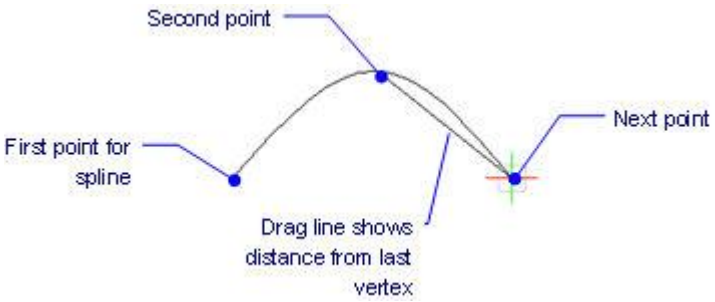
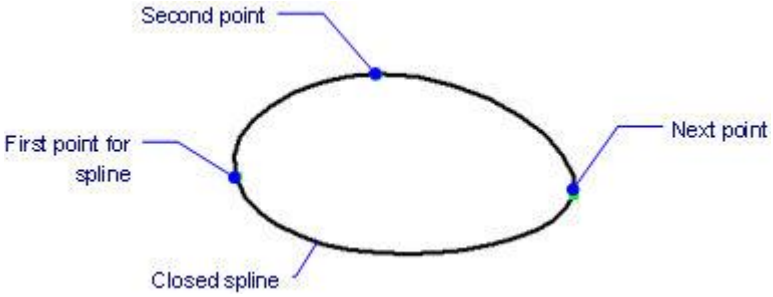
Set next point or [Close/Fit tolerance]: (Pick another point, or enter an option.)

Set next point or [Close/Fit tolerance]: (Pick another point, enter an option, or press Enter to end the spline.)

Select starting tangent point: (Pick a point, or press Enter for no starting tangent.)

Enter tangent for ending point: (Pick a point, or press Enter for no ending tangent.)

Command Options

Option	Description
First point for spline	Specifies the start of the tangent. Pick a point, or enter x,y coordinates.
Second point	Specifies the first vertex of the spline.
Next point	Specifies the next vertex.  This prompt repeats until you press Enter.
Close	Closes the spline; the end and start points are joined.  To open/close an existing spline: Select the spline, then edit the *Close* property in the Properties Bar.

Fit tolerance	Specifies how closely the spline matches its vertices (also known as "fit points"); prompts you: **Enter fit tolerance (in drawing units)** - enter 0 or a larger number. • 0 - spline passes through fit points. • More than 0 - spline passes further away from fit points.
Select starting tangent point	Locates a tangency for the starting point. Press Enter to ignore this option.
Enter tangent for ending point	Locates a tangency for the ending point. Press Enter to ignore this option.

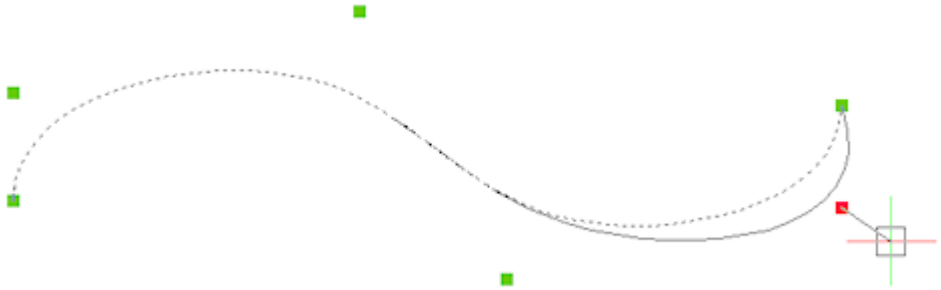
Procedures

Drawing splines

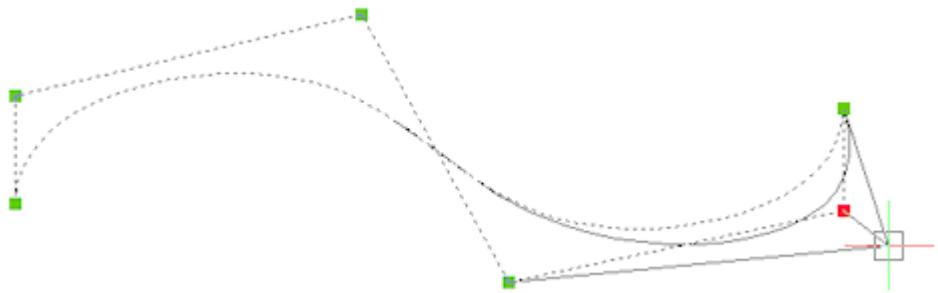
Grips Editing

Open and closed splines can be edited directly through grips. Splines can be grip edited either with fit points or control vertices. Choose either *Fit* or *Control vertices* from the *Method* property options of a spline in the Properties Bar. The *CV Frame* property of the spline controls whether the control vertices frame displays or not.

Method = Fit



Method = Control Vertices; CV Frame = Off



Method = Control Vertices; CV Frame = ON

1. Select the spline. Notice that it has one grip at each fit point or control vertex.
2. Drag a grip to stretch the spline.

Related Command

PEdit - turns polylines into splines.
Splinedit - modifies the properties of a spline or converses a spline to a polyline.
Helix - Creates a 2D spiral or a 3D helix.

Splinedit

Modifies the properties of a spline or converses a spline to a polyline.

Accessing the Command

command bar: *splinedit*
menu bar: *Menu| command*
toolbar: *Modify |*

: ***splinedite***
Prompts you in the command bar:

Select spline: (Select a spline entity.)

Edit spline [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: (Select an option.)

Edit spline [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: (Select an option or press Enter to conclude.)

Command Options

Option	Description
Close/Open	Depending on whether the selected spline is open or closed, one of the options is available: • Close: Closes an open spline connection the start and endpoint. • Open: Opens a closed spline by removing the segment between the first and last points specified when the spline was originally created.
Join	Joins the selected spline with a linear entity at coincident endpoints. Lines, polylines, arcs, elliptical arcs and splines are accepted; prompts you: Select any open curves to join to source - select a connected entity. Entities in set: 1 Select any open curves to join to source - select another connected entity or press Enter. Edit spline [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit> - press Enter to exit the command or choose another option to accept the join.
Fit Data	Edits fit point data using several options; prompts you: Edit fit data [Add/Close/Delete/Kink/Move/Purge/Tangents/toLerance/eXit] <eXit> - choose an option. Add: Adds fit point to the spline; prompts you: • Select existing fit point <exit> - select a fit point. The selected and the next fit point highlight. • Specify new fit point to add <exit> - pick a point. • Specify new fit point to add <exit> - pick a point or press Enter. • Select existing fit point <exit> - select a fit point or press Enter. Close/Open: • Closes an open spline connection the start and endpoint. • Opens a closed spline by removing the segment between the first and last points specified when the spline was originally created. Delete: Removes fit points; prompts you: • Select fit point to delete <exit> - select a fit point. • Select fit point to delete <exit> - select a fit point or press Enter.

	Kink: Adds a knot and fit point at the specified location on the spline, which does not maintain tangent or curvature continuity at that point; prompts you: • Specify a point on the spline <exit> - pick a point on the spline. • Specify a point on the spline <exit> - pick a point on the spline or press Enter. Move: Moves fit points one by one; prompts you: • Enter new location or [Next/Previous/Select point/eXit] <N> - pick a point, choose an option or press Enter to move the next fit point. • Enter new location or [Next/Previous/Select point/eXit] <N> - pick a point, choose an option, press Enter to move the next fit point or choose Exit to stop moving fit points. Purge: Replaces the fit data from the spline with control vertices. Tangents: Changes the starting and ending tangents of a spline; prompts you: • Specify start tangent or [System default] <System default> - pick a point or press Enter to accept the system default. • Specify end tangent or [System default] <System default> - pick a point or press Enter to accept the system default. • For closed splines you are prompted to define the tangent at the point of closure: Specify tangent or [System default] <System default> - pick a point or press Enter to accept the system default. **Tolerance**: Refits the selected spline to the existing fit points using a new tolerance value; prompts you: Enter fit tolerance <0.000000> - type value. Exit: Accepts the changes and returns to the previous prompt.
Edit Vertex	Allows to edit control frame date using several options; prompts you: Edit vertex [Add/Delete/Elevate order/Move/Weight/eXit] <eXit> - choose an option. Add: Adds a new control vertex at the point you specify. • Specify a point on the spline <exit> - pick a point on the spline. • Specify a point on the spline <exit> - pick a point on the spline or press Enter. Delete: Removes control vertices. • Select control point to delete <exit> - select a vertex. • Select control point to delete <exit> - select a vertex or press Enter. Elevate Order: Increases the polynomial order (degree plus one) of the spline. This results in increasing the number of control vertices across the spline. The maximum value is 26; prompts you: Enter new order <xx> - type an integer. Move: Relocates control vertices one by one; prompts you: • Enter new location or [Next/Previous/Select point/eXit] <N> - pick a point, choose an option or press Enter to move the next fit point. • Enter new location or [Next/Previous/Select point/eXit] <N> - pick a point, choose an option or press Enter to move the next fit point or press Enter to stop moving vertices. Weight: Changes the weight of a specified control vertex. Recalculates the spline based on the new weight value for the specified control vertex. A larger value pulls the spline closer to the control vertex; prompts you: • Enter new weight (current = 1.000000) or [Next/Previous/Select point/eXit] <N>:Enter new weight (current = 1.000000) or

	[Next/Previous/Select point/eXit] <N> - enter a value or choose an option. • Enter new weight (current = 1.000000) or [Next/Previous/Select point/eXit] <N>:Enter new weight (current = 1.000000) or [Next/Previous/Select point/eXit] <N> - enter a value or choose an option or press Enter. Exit: Accepts the changes and returns to the previous prompt.
Convert to Polyline	Allows to convert a spline to a polyline. The precision value determines how closely the resulting polyline matches the spline: higher values create more segments. Values between 0 and 99 are accepted; prompts you: Specify a precision <10> - enter an integer. The PLINECONVERTMODE system variable specifies how to convert a spline to a polyline: • 0: Create polylines with line segments. • 1: Create polylines with arc segments.
Reverse	Reverses the direction of the spline.
Undo	Undoes the previous modification.
Exit	Concludes the editing of the spline.

Related Commands

Join - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.
PEdit - edits polylines, 3D polylines, and 3D meshes.
Spline - draws open and closed splines.

Spotlight

Creates a spot light.

Accessing the Command

command bar: *spotlight*
menu bar: *View | Rendering | Lights | New spot light*
toolbar: *Lights |*

: ***spotlight***
Prompts you in the command bar:

Specify source position <0,0,0>: (specify the position of the light.)

Specify target position <0,0,-10>: (specify the target point of the light.)

If the LIGHTINGUNITS system variable is 0 (No lighting units), the following prompt displays:

Enter an option to change [Name/Intensity/Status/Hotspot/Falloff/shadoW/Attenuation/Color/eXit] <eXit>:

If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), the following prompt displays:

Enter an option to change [Name/Intensity factor/Status/Photometry/Hotspot/Falloff/shadoW/Attenuation/filterColor/eXit] <eXit>:

Command Options

Option	Description
Name	Enter light name <Spotlight1>: (Enter a name or press Enter to accept the default name.)
Intensity Factor Intensity	Enter intensity (0.00 - max float) <1.0>: (Enter the intensity. Values between 0.00 and the maximum value supported by your system are accepted.)
Status	Enter status [ON/OFf] <ON>: (Choose ON to include the light when calculating a rendering.)
Hotspot Falloff	Enter hotspot angle (0.00-160.00) <45.0>: (Enter a value between 0.00 degrees and 160.00 degrees). Enter falloff angle (0.00-160.00) <50.0>: (Enter a value between 0.00 degrees and 160.00 degrees). If yout enter a Falloff angle that is smaller than the Hotspot angle, the latter is set equal to the Falloff angle. When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot) that is surrounded by an area of lesser intensity (falloff). • Hotspot angle: angle of the central light cone (defines the hotspot) • Falloff angle: angle of the full light cone The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity. If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp. The greater the difference between both angles, the softer the edge of the light cone.
Photometry	**Enter a photometric option to change [Intensity/Color/eXit] <Intensity>:**

- Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>: (Enter the intensity expressed in Candela (Cd).)
    - Enter Flux (Lm) : (Enter the luminous flux expressed in Lumen (Lm).)
    - Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>: (Enter the illuminance expressed in Lux (Lx) or Foot-candle (Fc).)
        - Enter Distance <1.0>: (Enter a distance expressed in drawing units.)
- Enter color name or enter an option [?/Kelvin] <D65>:
    - Enter color name(s) to list <*>: (Enter a color name from the list. Type * to see the list)
        - D65
        - Fluorescent
        - Coolwhite
        - Whitefluorescent
        - daylightfluorescenT
        - Incandescent
        - Xenon
        - Halogen
        - Quartz
        - Metalhalide
        - mErcury
        - Phosphormercury
        - highpressureSodium
        - Lowpressuresodium
    - Enter Kelvin temperature <3600.0>: (Enter a color temperature expressed in degrees Kelvin (K).)

Candela: The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela.

Lumen: The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr

As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens.

Illuminance: In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human brightness perception.

	If LIGHTINGUNITS=2 (International lighting units): In SI derived units illuminance is measured in lux (lx) or lumens per square metre. If LIGHTINGUNITS=1 (American lighting units): The foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux, although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux. Kelvin temperature: Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K). Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700&endash;3,000 K) are called warm colors (yellowish white through red).

Shadow	**Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled]: <Sharp>:**
	• Off: Disables the calculation of shadows for the light.
	• Sharp: Displays shadows with sharp edges. Use this option to increase performance.
	• Soft mapped: Displays realistic shadows with soft edges.
	Prompts you: Enter map size [64/128/256/512/1024/2048/4096] <256>:
	• Soft sampled: Displays realistic shadows with softer shadows based on extended light sources.
	Prompts you: Enter an option to change Shape/sAmples/Visible/<eXit>:
	  • Enter shape [Disk/Rect] <Disk>:
	  • Enter radius <0.0>: (Enter the radius of the disk.)
	  • Enter Length <0.0>: (Enter length of the rectangle.)
	  • Enter Width <0.0>: (Enter width of the rectangle.)
	  • Enter Shadow Sample <16>:
	  • Enter Shape Visibility [Yes/No] <No>:

Attenuation	Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit] <eXit>:
	Enter an option or press Enter to return to the base options.
	  • Enter attenuation type [None/Inverse linear/inverse Squared] <None>:
	  • None: No attenuation. The distance to the light source has no influence.
	  • Inverse linear: The attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.

	• Inverse squared: The attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong. • Specify start limit offset <1.0>: Define the point where the light starts to shine, measured from the center of the light. • Specify end limit offset <10.0>: Define the point where the light stops to shine, measured from the center of the light.
Filter Color Color	**Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>:** • True Color (RGB): The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • Index color: Specifies an AutoCAD Color Index color. • HSL: Specifies the color using three parameters Hue, Saturation and Lightness.
Exit	Concludes the command and creates the light.

Related Commands

Distantlight - creates a distant light.
Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Pointlight - creates a point light.
Render - calculates a photorealistic image of the 3D model.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.
Weblight - creates a web light.

Start

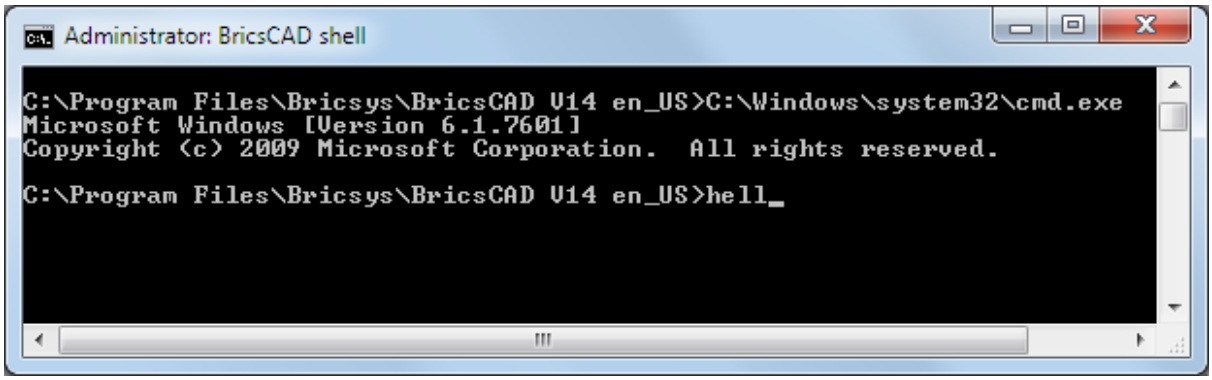
Runs Operating Sustem applications.

Accessing the Command

command bar: *start*
transparent: *'start*

: ***start***

Prompts you in the command bar:
Application to start: *(Enter the name of a Windows program, or press Enter for the command prompt window.)*



Enter commands or names of programs, such as 'Notepad'.

For a list of operating system commands, enter Help.

To exit the window, enter Exit.

Command Options

Option	Description
Application to start	Specifies the name of the Windows program to run. Press Enter to open the Windows command prompt window.

Related Commands

Shell - opens the operating system command prompt window; runs other applications.
Sh - opens the operating system command prompt window; runs other applications.
Files - displays the operating system's File Manager.

StatBar

Toggles the display of the status bar (short for "status bar").
Sets the value of the WNDLSTAT system variable.

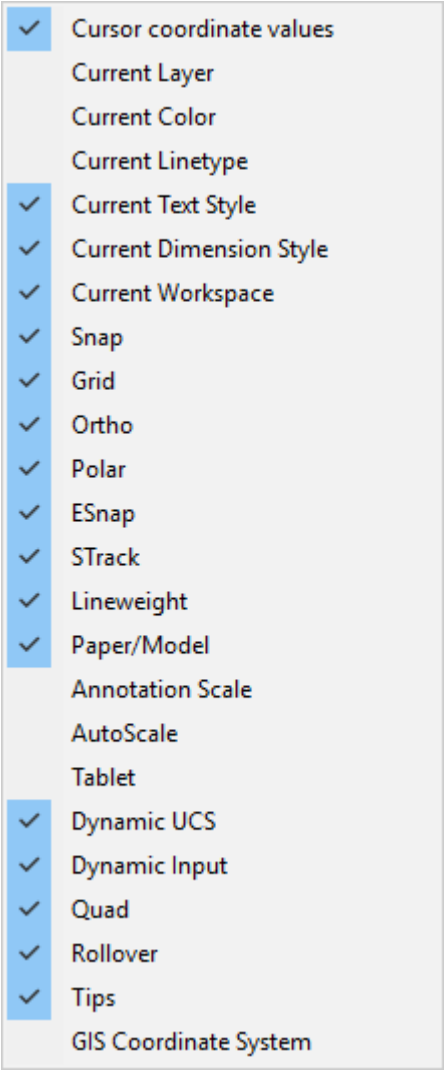
Accessing the Command

command bar: *statbar*
transparent: *'statbar*
shortcut: *Shift + F3*
menu bar: *View | Status Bar*

: ***statbar***
Prompts you in the command bar:
WNDLSTAT is currently on [ON/OFf/Toggle] <On>: (Enter an option.)

The status bar provides useful information and controls:

1. Status field:
   2. displays a tooltip when hovering over a tool button
   3. displays command prompts when the command bar is closed
   4. double click the status field to toggle the command bar on/off
2. Coordinates field
3. Drawing properties
4. Drawing aid fields
5. Status bar option button.
   Displays a list: Click an option to toggle the corresponding field on/off.



The only reason to turn off the status bar is to gain a bit more drawing area.

Procedures

Working with the status bar

Related Commands

CleanScreenOff - displays user interface elements that were hidden by te CleanScreenOn command.

CleanScreenOn - hides elements of the user interface.

CommandLine - shows the command line.

CommandLineHide - hides the command line.

GraphScr - brings the graphics window to the foreground.

Scrollbar - toggles the display of the scroll bars.

TextScr - brings the text window to the foreground.

Toolbar and -Toolbar - toggle the display of toolbars.

Status

Reports the status of the drawing.

Accessing the Command

command bar: *status*
transparent: *'status*
menu bar: *Tools | Drawing Status*
toolbar: *Inquiry |*

: ***status***

Displays the status report in the command bar.

To view the full report, press **F2** to open the Prompt History window.

A typical report looks like this:

Status:

```
   Current drawing name:  lsedit.dwg
     Drawing limits are:  X=   0.0000  Y=   0.0000
                          X=  12.0000  Y=   9.0000
  Paper space limits are:  X=   0.0000  Y=   0.0000
                          X=  12.0000  Y=   9.0000
  Screen width  (pixels):  909
  Screen height (pixels):  497
      Insertion base is:  X=   0.0000  Y=   0.0000  Z=   0.0000
     Snap resolution is:  X=   0.5000  Y=   0.5000
       Grid spacing is:  X=   0.5000  Y=   0.5000
         Current layer:  0
         Current color:  BYLAYER
      Current linetype:  ByLayer
     Current elevation:  0.0000
     Current thickness:  0.0000
                  Fill:  on
                  Grid:  off
                 Ortho:  off
                  Snap:  off
                 Blips:  off
                  Drag:  on
          Command echo:  on
Positive angle direction:  Counterclockwise
         Angular units:  Decimal degrees
       Dimension units:  Decimal
        Pickbox height:  3
    Entities in drawing:  136
```

Command Options

No options.

StlOut

Exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing).

Accessing the Command

command bar: *stlout*

: ***stlout***

Prompts you in the command bar:

Select 3d solids or meshes to export: (Select solids and/or meshes. Right click to stop selecting.)

Create binary STL file? <Yes>/No: (Select Yes or No.)
Displays a Save File dialog
(Type a name for the file and click the Save button.)

Command Options

Option	Description
Create binary STL file?	Yes: Creates an STL file. No: Creates an ASCII text file.
System Variables and User Preferences	When exporting 3d solids to STL, curved faces will indeed become faceted. The default way to control the facet resolution (= the smoothness) is to use the FACETRES system variable: default value is 0.5, minimum = 0.01 (coarse) - maximum = 10 (very fine) Using MODELERPROPERTIES it is possible to override Facetres and control the smoothness by using different parameters, but in general it is recommended to use facetres because it is simpler.

Related Commands

Export - Exports drawings in a variety of vector and raster formats.
-ModelerProperties - controls the various settings of the Acis modeler through the command bar.
ModelerProperties - controls the various settings of the Acis modeler through the *Settings* dialog box.

StopScript

Stops the recording of scripts begun by the RecScript command.

Accessing the Command

command bar: *stopscript*
menu bar: *Tools | Stop Recording*
toolbar: *Tools |*

: ***stopscript***

Displays no prompt in the command bar.

Use the **RecScript** command to start recording scripts.

For more information writing scripts, see 'Customizing BricsCAD.'

Command Options

No options.

Related Commands

RecScript - records scripts to SCR files.
Script - loads and runs a SCR script file.
RScript - reruns the script.
Delay - pauses the execution of the script for the specified number of milliseconds.
Resume - resumes a stopped script.

Stretch

Stretches entities.

Accessing the Command

command bar: *stretch*
alias: *s*
menu bar: *Modify | Stretch*
toolbar: *Modify |*

: ***stretch***
Prompts you in the command bar:

Select entities to stretch by crossing-window or crossing-polygon: (Pick a point, or enter C or CP.)

Opposite Corner: (Pick another point.)

This command works only with crossing selection modes.

Entities in set: 1

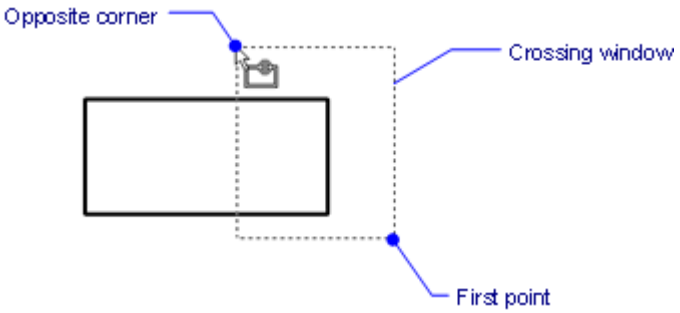
Select entities to stretch by crossing-window or crossing-polygon: (Choose more entities, or press Enter to continue.)

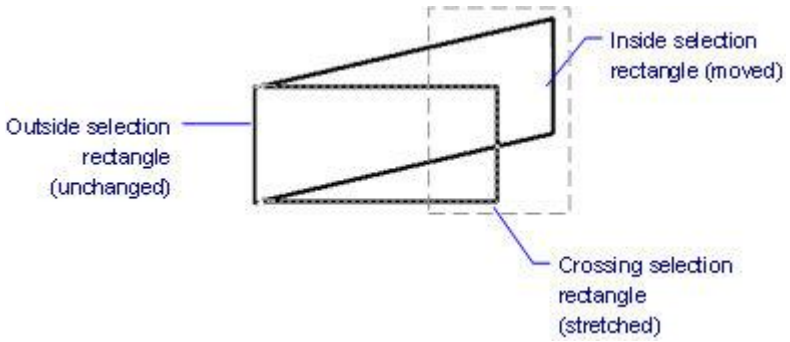
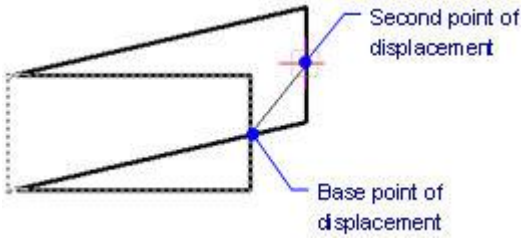
Base point of displacement: (Pick a point, or enter x,y coordinates.)

Second point of displacement: (Pick another point.)

Text cannot be stretched.

Command Options

Option	Description
Select entities	Selects the entities to stretch; must use Crossing or Crossing Polygon mode: • C - crossing mode. • CP - crossing polygon mode. When using the cursor, pick the second point to the left of the first:  Crossing mode is indicated by a dashed selection rectangle. The selection rectangle (or polygon) determines how this command affects entities: • Entities crossing the selection rectangle are stretched. • Entities within the rectangle are moved. • Entities outside the rectangle remain in place.

	
Base point of displacement	Specifies the starting point of the stretch.
Second point of displacement	Specifies the stretch distance by the distance between two points, base and second point. 

Procedures

To stretch entities
To stretch entities using grips

Related Commands

Lengthen - changes the length of open entities.
Extend - extends open entities.
Scale - changes the size of entities.
Properties - changes the geometry of entities.

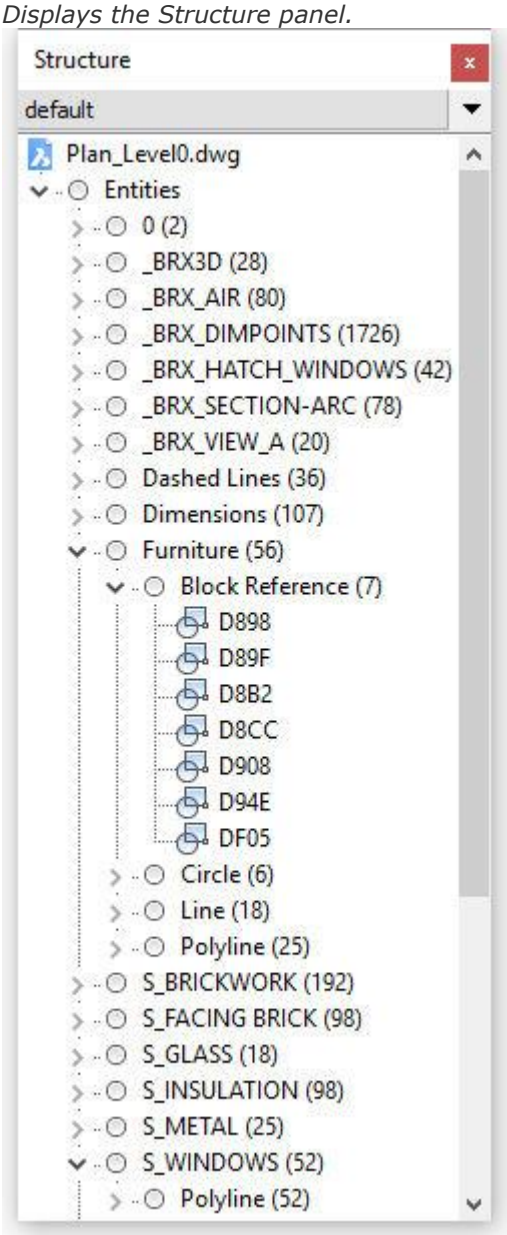
StructurePanel

Opens the Structure panel.

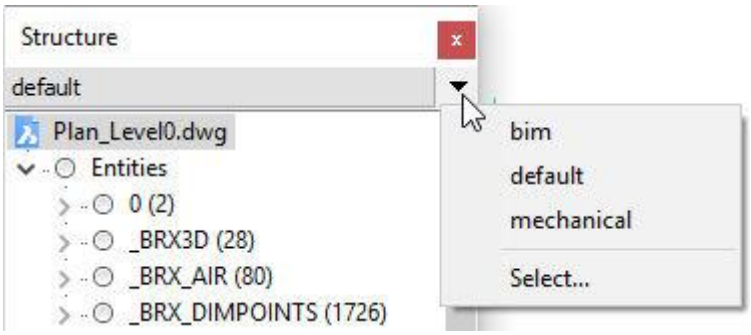
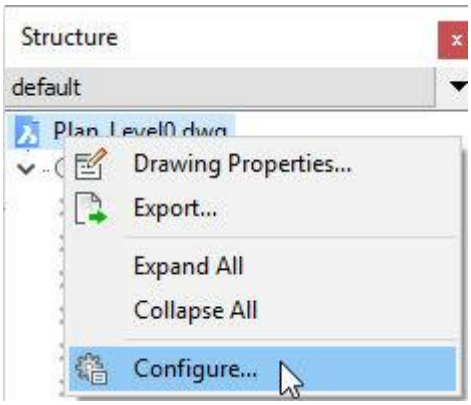
Accessing the Command

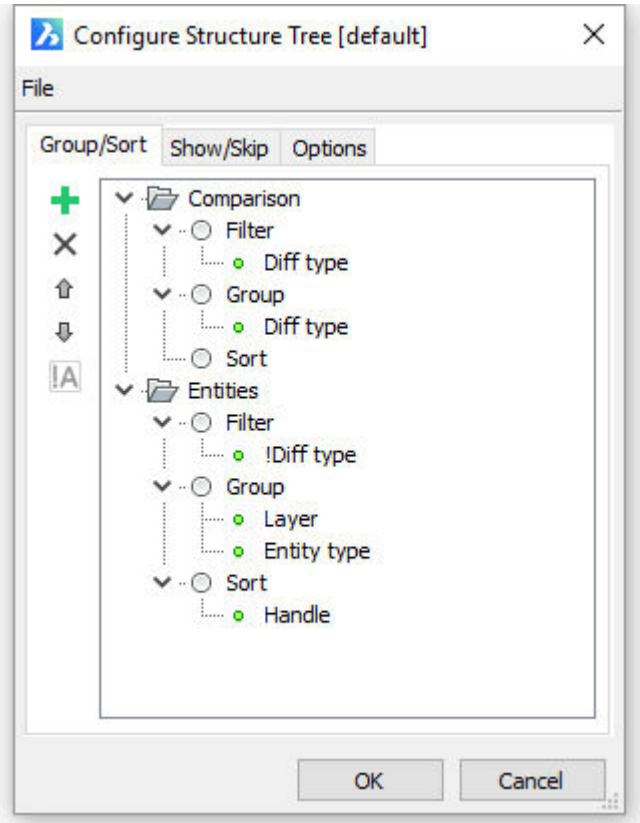
command bar: *structurepanel*
context menu: *Structure*

: ***structurepanel***
Displays the Structure panel.



Command Options

Option	Description
Select a configuration	Loads a *Structure Tree Configuration File* (*.cst) Click the down arrow button.  Lists the *.cst configuration files in the Support folder (e.g. C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support) Click *Select...* to search for *.cst configuration files in another folder.
Edit the configuration	Allows to control composition of the structure tree. Right click the drawing name, then choose *Configure...* in the context menu.  The *Configure Structure Tree* dialog opens:

	
Export the structure	Exports the current structure to an *.xml file Right click the drawing name, then choose *Export...* in the context menu.

Procedures

To open the Structure panel
To configure the tree-structure on the Structure panel
To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

StructurePanelClose - closes the Structure panel.

StructurePanelClose

Closes the Structure panel.

Accessing the Command

command bar: *structurepanelclose*

: ***structurepanelclose***
The Structure panel is closed. If the Structure panel is currently docked, the Structure Panel tab is removed.

Command Options

No command options.

Related Commands

StructurePanel - opens the Structure panel.

Style

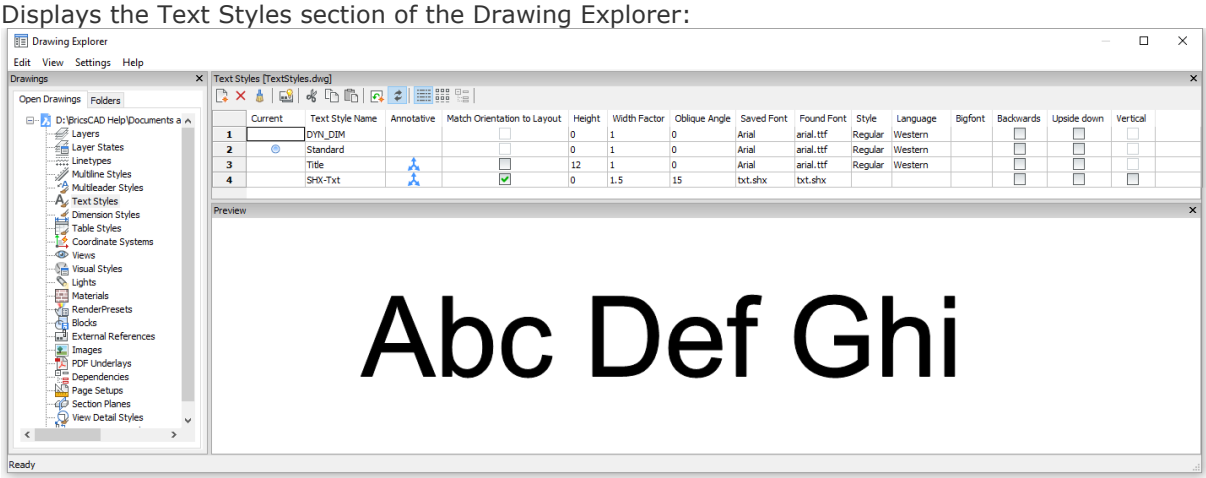
Creates and modifies text style with the Drawing Explorer.

Accessing the Command

command bar: *style*
aliases: *ddstyle, expfonts, expstyle, expstyles, st*
menu bar: *Tools | Drawing Explorer | Text Styles*
status bar: *Standard*

: ***styles***

Displays the Text Styles section of the Drawing Explorer:

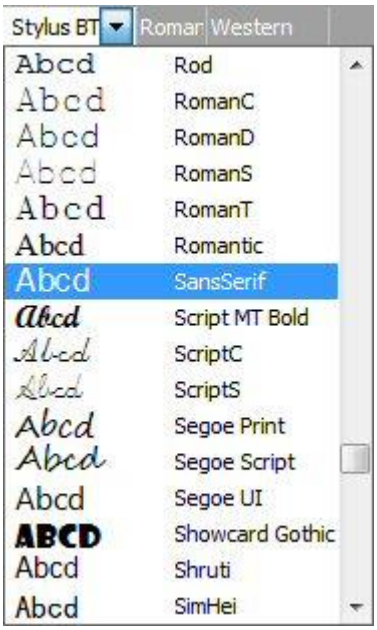


Press **F1** to access help.

Make changes to styles, or create new ones, and then click X to exit.

Command Options

Option	Description
Current	Designates the current text style. This is the style used by default in the **Text** and **MText** commands.
Text Style Name	Specifies the style name: • To rename, click the name twice, and then rename. • To create a new text style, click the New button to create a new text style.
Annotative	Sets the Annotative property of the text entities created with this style. In an annotative text style the Height defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.
Match Orientation	Matches the orientation of the text to the orientation of the paperspace layout.
Height	Specifies the height of the text. Enter 0 or a number larger than 0: • 0 - does not set the height; TEXT command prompts for height. • Not 0 - specifies the height of text; TEXT command does not ask for height.

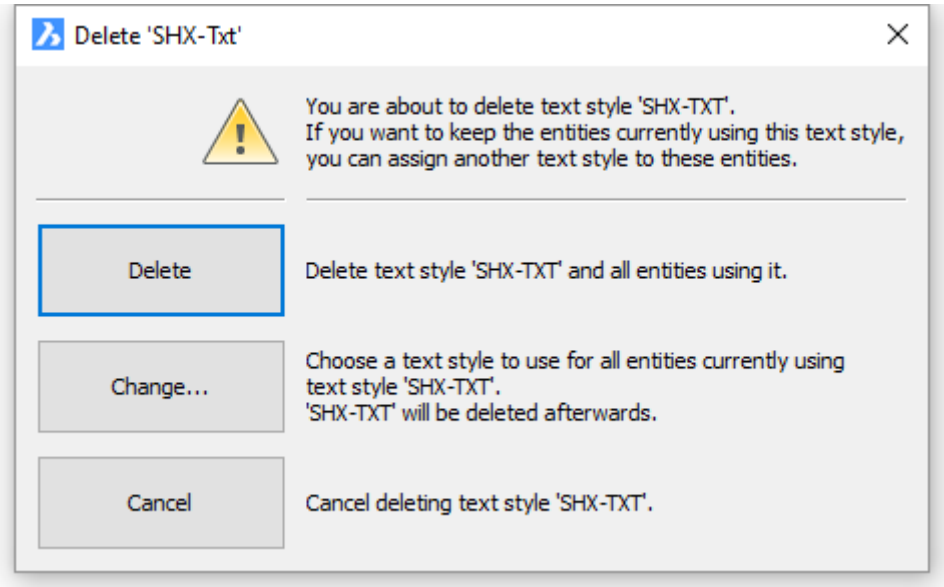
Width Factor	Specifies the width factor of the text. For example, entering 2 makes the text twice as wide. Enter a number larger or smaller than 1: • Less than 1 - text is squeezed narrower. Width factor = 0.5 • More than 1 - text is spread wider. Width factor = 2
Oblique Angle	Specifies the obliquing angle for the text. For example, entering 15 slants the text forward by 15 degrees. Enter a positive or negative number: • Negative number - text slants to the left. Oblique angle = -15 • 0 - text is upright. • Positive number - text slants to the right. Oblique angle = 15
Saved Font	Specifies the SHX or TTF font file to use; choose a name from the droplist:  • SHX are compile shape files created for AutoCAD and BricsCAD. • TTF are TrueType font files created for Windows and Macintosh. BricsCAD can use any SHX and TTF font found on your computer.
Found Font	Shows the name of the font file found. If the font defined in by *Saved Font* is not found .shx fonts are replaced by the font defined by the FONTALT (Alternative font) system variable. TTF fonts are replaced according to the replacement table defined by the FONTMAP system variable. By default this is *Default.fmp* in the *Support* subfolder of the roamable root folder (e.g. C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V18x64\en_US\Support

	Fonts that have been replaced display in Red. Saved Font / Found Font Arial / arial.ttf Arial / arial.ttf romans.shx / simplex.shx Arial / arial.ttf
Style	Specifies style for TTF fonts; choose a style from the droplist: Style: Regular Bold Bold Italic Italic Regular • Regular - applies neither bold nor italics. • Bold - applies boldface. • Italic- applies italics. • Bold Italic - applies bold and italics. This property is not available for SHX fonts.
Language	Specifies the language for TTF fonts; choose one from the droplist: Language: Western Arabic Baltic Central Europ Cyrillic Greek Hebrew Turkish Vietnamese Western Available languages may vary for each font. This property affects accents and other attributes specific to languages. This property is not available for SHX fonts; use Bigfont property instead.
Bigfont	Lists the names of SHX bigfont files; choose one from the droplist: Bigfont / Backwards / Upside d hangul.sh Abcd hangul.shx Abcd japanese.shx Abcd trad_chin.shx Abcd (none) "Bigfonts" is the name given to large SHX files that contain characters for multiple languages. This property is not available for TTF fonts; use Language property instead.

Backwards*	Toggles backwards text. Backwards text is useful when text is plotted on the back of mylar or other clear media. • Yes - text is drawn backwards. Backwards • No - text is drawn forwards.
Upside down*	Toggles upside-down text. • Yes - text is drawn upside down. upsidedown • No - text is drawn rightside up.
Vertical	Toggles vertical text. Only some SHX fonts support vertical text; this option is not available for TTF fonts. • Yes - draws text vertically. The Text command sets the rotation angle to 270 degrees automatically. Vertical • No - draws text horizontally.

* This setting applies to Text entities only, not to MText entities.

Toolbar Option	Description
New	Creates new text styles. The style is named "NewStyle1" and inherits the properties from the currently selected text style. To rename the style: 1. Click the name twice. 2. Enter the new name.
Delete	Erases the selected style. The current style and the *Standard* style cannot be deleted. If the style is in use, a dialog box displays:



Choose an option.

- Delete: *Caution!* Erases the style, as well as all text that uses the style!
- Change: Selects another style before erasing it. Entities using the doomed style are given the alternative that you select.
  Displays list of available styles:

  31
  32
  41
  42
  43
  STANDARD

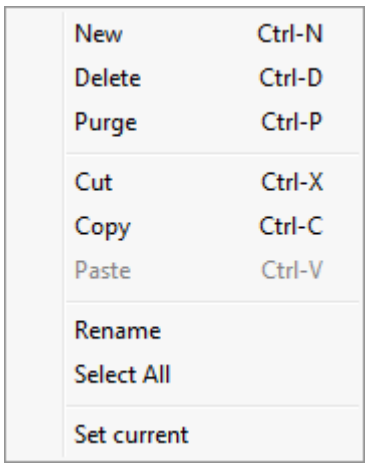
  1. Choose style name.
  2. Click *Change*.

  The style is erased and text takes on the selected style.
- Cancel: Cancels the operation; the style is not erased.

Purge	Removes unused styles from the drawing. The following styles cannot be purged: • Standard text style. • The current style. • Styles being used in the drawing.
Hide Xref Symbols On/Off	Allows to toggle the display of XRef text styles in the list.

Shortcut Menu Options

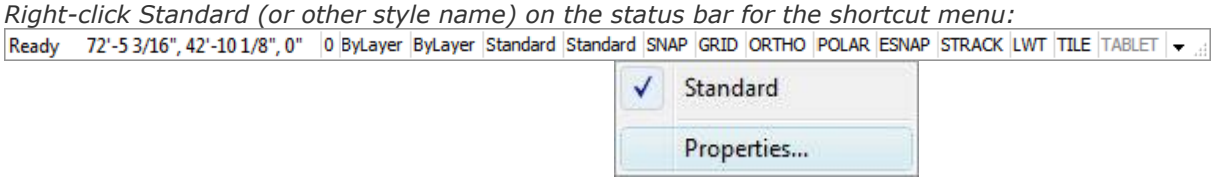
Right-click layer names in Drawing Explorer to see the following shortcut menu:



Option	Keystroke	Description
New	Ctrl+N	Creates new text styles.
Delete	Ctrl+D	Deletes styles from Drawing Explorer. The following styles cannot be deleted: • Standard. • Current style.
Purge	Ctrl+P	Purges styles from the current drawing. The following styles cannot be purged: • Standard. • Current style. • Any style used by text.
Cut	Ctrl+X	Cuts the style and its properties to the Clipboard.
Copy	Ctrl+C	Copies the style and its properties to the Clipboard.
Paste	Ctrl+V	Pastes the style from the Clipboard into Drawing Explorer.
Rename		Renames the selected style.
Select All		Selects all styles.
Set Current		Specifies the current working style. Only one style can be current at a time.

Status Bar Menu

Right-click Standard (or other style name) on the status bar for the shortcut menu:



Shortcut Menu	Description
Standard	Chooses the default text style. When the drawing contains additional text styles, their names are listed here.
Properties	Opens the Text Styles section of the Drawing Explorer.

Procedures

To open the Text Style Explorer dialog
To create a text style
To edit a text style
To make a text style current
To delete a text style

Related Commands

-Style - creates and modifies text styles through the command bar.
Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Change - changes the position of text.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Properties - modifies the properties of text.

Explorer - Opens the Drawing Explorer dialog box.

-Style

Creates and modifies text styles in the command bar.

Accessing the Command

command bar: *-style*
transparent: *'-style*
alias: *font*
status bar: *Standard*

: ***-style***
Prompts you in the command bar:
? to list existing styles/<Text style to create or modify> <STANDARD>: (Enter a name, or type ?)

Font <txt.shx>: (Enter the name of a SHX or TTF font file, or press Enter to accept the default value.)

font: txt.shx bigfont: is selected

Text height <0">: (Enter a number, or press Enter to accept the default value.)

Width factor <1.0000>: (Enter a number, or press Enter to accept the default value.)

Obliquing angle <0">: (Enter a number, or press Enter to accept the default value.)

Draw text backwards? <No> (Type Y or N.)

Draw text upside-down? <No> (Type Y or N.)

Draw text vertically? <No> (Type Y or N.)

This command is used to create styles used by the Text command.

Styles are also used by the MText command, but it can override all properties set by -Style.

Command Options

Option	Description
Text style to create or modify	Specifies the style name: • To modify an existing style, enter its name. • To create a new style, enter a new name.
? to list existing styles	Lists style names in the drawing; prompts you: Text style(s) to list - specifies which styles to list: • * - lists all style names. • name* - lists all style names beginning with name. • name - lists the style named name. The listing looks like this: **Text styles:** **Style name: STANDARD** **Font file: txt.shx** **Height: 0" Width factor: 1.000 Obliquing angle: 0"** **Generation: Normal**

Font	Specifies the SHX or TTF font file to use: • SHX are font files created for AutoCAD and BricsCAD. • TTF are font files created for Windows and Macintosh. BricsCAD can use any SHX and TTF font found on your computer.
Text height	Specifies the height of the text. Enter 0 or a number larger than 0: • 0 - does not set the height of text; Text command prompts for height. • Not 0 - sets the text height; 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 - squeezes text narrower. Width factor = 0.5 • More than 1 - spreads text wider. Width factor = 2
Obliquing angle	Specifies the obliquing angle for the text. For example, entering 15 slants the text forward by 15 degrees. Enter a positive or negative number: • Negative number - slants text to the left. Oblique angle = -15 • 0 - text is upright. • Positive number - slants text to the right. Oblique angle = 15
Draw text backwards?	Toggles backwards text. Backwards text is useful when text is plotted on the back of mylar or other clear media. Enter Y or N: • Yes - draws text backwards. Backwards • No - draws text forwards.
Draw text upside-down?	Toggles upside-down text. Enter Y or N: • Yes - draws text upside down. upsidedown • No - draws text rightside up.

Draw text vertically?	Toggles vertical text. Only some SHX fonts support vertical text; TTF fonts cannot be drawn vertically. This prompt is not displayed for TTF fonts. The Text command sets the rotation angle to 270 degrees. Enter Y or N: • Yes - draws text vertically. • No - draws text horizontally.

Status Bar Menu

Right-click Standard (or other style name) on the status bar for the shortcut menu:

Shortcut Menu	Description
Standard	Chooses the default text style. When the drawing contains additional text styles, their names are listed here.
Properties	Opens the Text Styles section of the Drawing Explorer. See Style command.

Procedures

To open the Text Style Explorer dialog
To create a text style
To edit a text style
To make a text style current
To delete a text style

Related Commands

Style - creates and modifies text styles through the Drawing Explorer.
Text - places single-lines of text in drawings.
-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Change - changes the position of text.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Properties - modifies the properties of text.

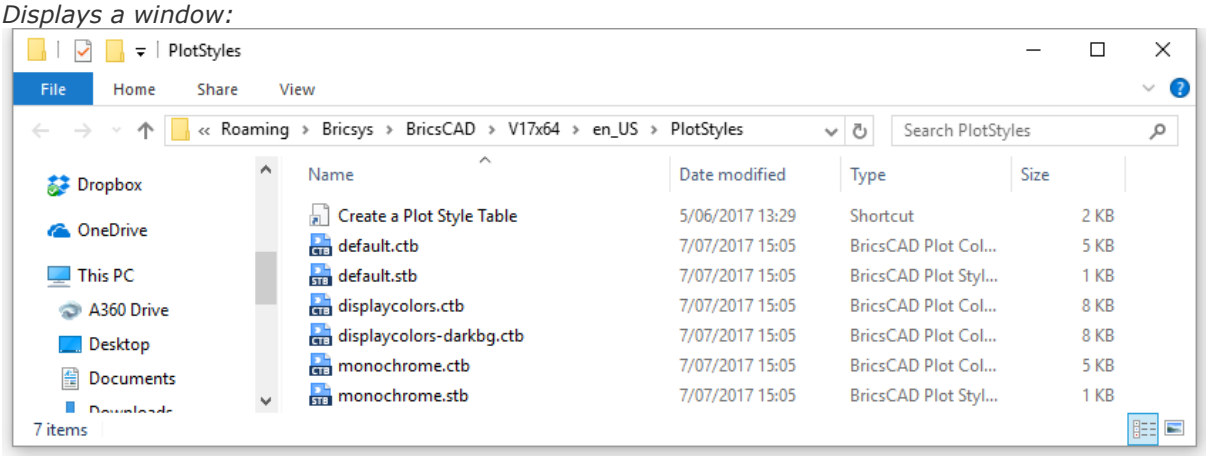
StylesManager

Creates and attaches plot style files.

Accessing the Command

command bar: *stylesmanager*
menu bar: *File | Styles Manager*

: ***stylesmanager***
Displays a window:

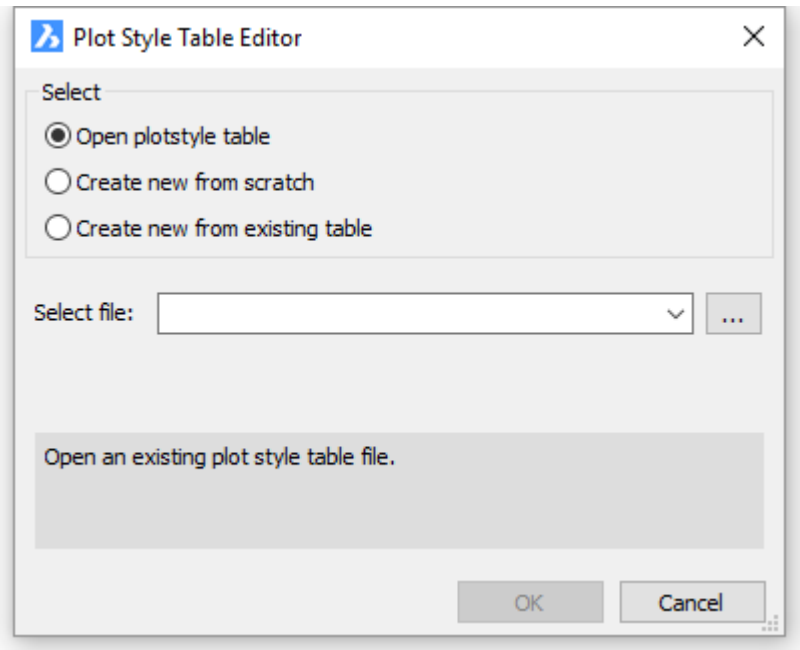
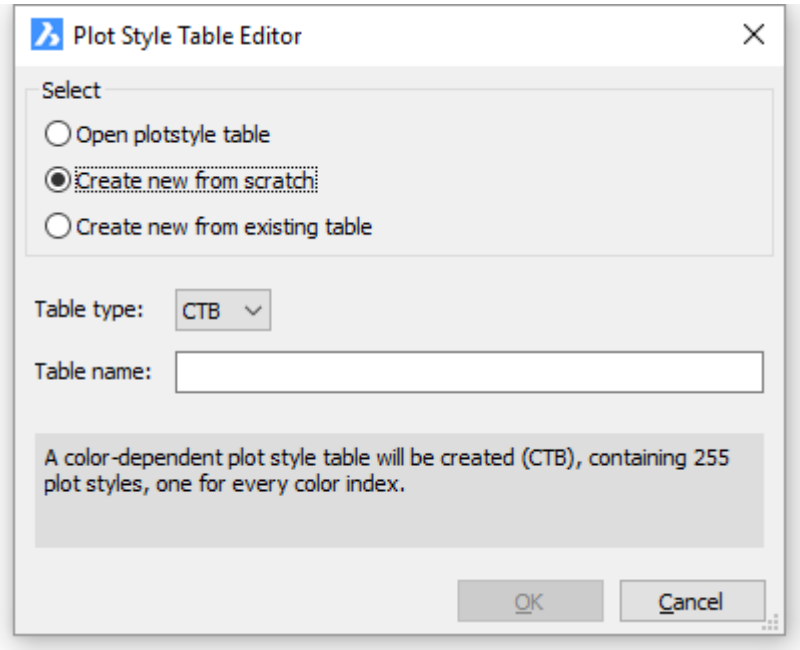


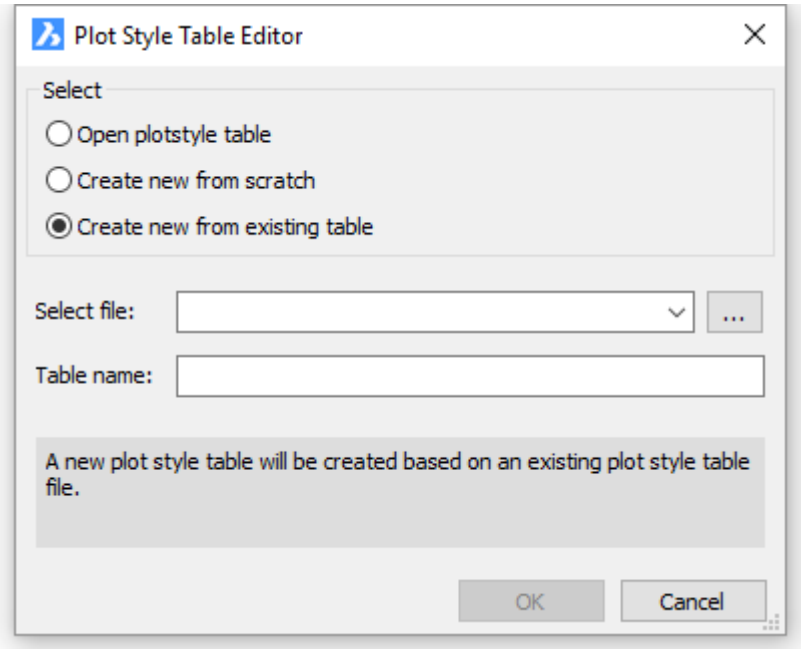
Double-click a CTB or STB file to open the Plot Style Table Editor. See **PlotStyle** command.
CTB and STB files control how drawings are printed:

- CTB files use entity colors to assign printing properties.
- STB use plot styles to assign printing properties.

Double-click Create a Plot Style Table to launch the Add Plot Style Table wizard.

Command Options

Option	Description
	Open plot style table - opens an existing plot style table Click the *Browse* button (...), then choose an existing CTB or STB file. Click the *OK* button to open the Plot Style Table Editor dialog.
	**Create from scratch:** Select a table type: • CTB (Color-dependent Plot Style Table) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file. • STB (Named Plot Style Table) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.

	Type a name in the *Table name* field. Click the *OK* button to open the Plot Style Table Editor dialog.
	Create new from existing table - A new plot style table will be created based on an existing plot style table. Select an existing file from the *Select file* drop down list or click the *Browse* button (...), then choose an existing CTB or STB file and click the *OK* button. Type a name in the *Table name* field. Click the *OK* button to open the Plot Style Table Editor dialog.

Procedures

To open the Plot Styles Manager
To create a new plot style table
To create a new plot style table in the Print dialog
To edit a plot style table
To edit a plot style table in the Print dialog

Related Commands

ConvertCtb - converts CTB plot style files to STB files.

ConvertPStyles - converts drawings from CTB to STB plot styles.

Layer - assigns plot styles to layers.

PageSetup - assigns plot style to drawings.

PlotStyle - sets the current plot style.
Print - plots drawings with plot styles.
Properties - changes plot styles assigned to entities.

Subtract

Performs Boolean subtraction operations on 3D solids and 2D regions; removes one from the other.

Accessing the Command

command bar: *subtract*
alias: *su*
menu bar: *Model | 3D Solid Editing | Subtract*
toolbar: *Solid Editing |*

: ***subtract***
Prompts you in the command bar:

Select ACIS object to subtract from: (Choose the first set of one or more 3D solids and/or 2D regions.)

Entities in set: 1

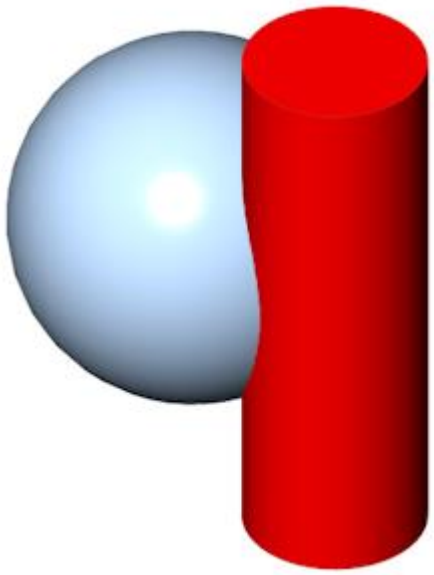
Select ACIS object to subtract from: (Choose more ACIS entities, or press Enter to continue.)

Select ACIS objects to subtract: (Choose the second set of one or more 3D solids and/or 2D regions.)

Entities in set: 1

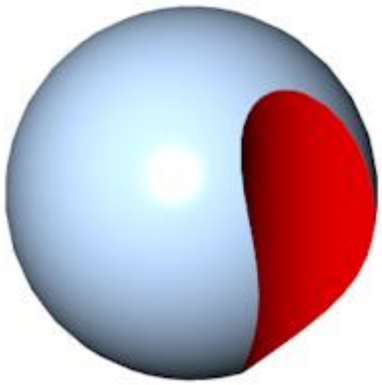
Select ACIS objects to subtract: (Choose more ACIS entities, or press Enter to continue.)

The DELETETOOL system variable controls whether the subtracted entities (tools) are deleted or retained.

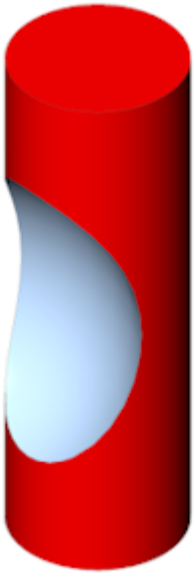


The result depends on the order in which ACIS entities are selected.

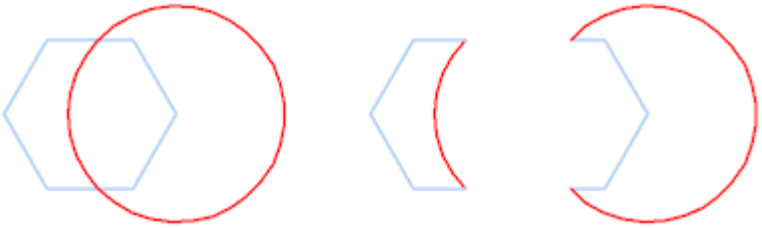
- Select the sphere, and then subtract the cylinder:



- Select the cylinder, and then subtract the sphere:



This command also applies to regions:



Command Options

Option	Description
Select ACIS object to subtract from	Specifies the ACIS entities that will have a portion removed. Choose the first set of one or more 3D solids and/or 2D regions.
Select ACIS objects to subtract	Specifies the ACIS entities that will be removed. Choose the second set of one or more 3D solids and/or 2D regions.

Related Commands

Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Union - performs Boolean union operations on 3D solids and 2D regions; adds one to another to form a single entity.

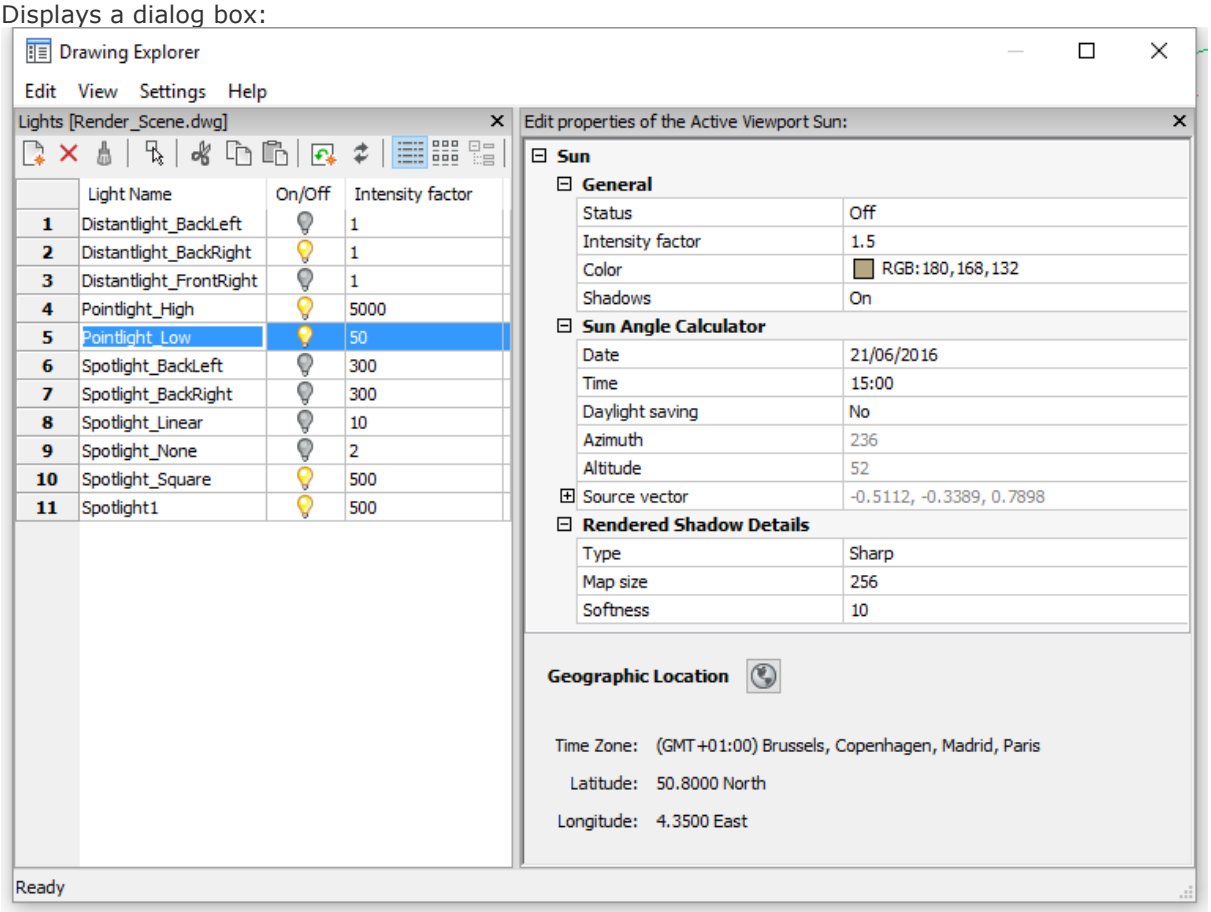
Sunproperties

Displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.

Accessing the Command

command bar: *sunproperties*
alias: *sun*
menu bar: *Tools | Drawing Explorer | Lights* and *View | Rendering | Lights | Sun Properties...*
toolbar: *Lights |*

:sunproperties
Displays a dialog box:



Settings

Setting	Description
General	
Status	Turns the sun Off/On. If turned off, the sun properties have no effect.
Intensity factor	Defines the intensity or brightness of the sun. The higher the number, the brighter the light.
Color	Sets the color of the sunlight. Select a color in the drop-down list or choose *Select Color* to open the Select Color dialog box. When LIGHTINGUNITS = 1 or 2, the *Color* property of the sun cannot be edited.

Shadows	Switches the calculation and display of shadows for the sunlight. Turn this option off to increase performance.
Sun Angle Calculator	
Date	Displays the current date. Click the field to adjust the date.
Time	Displays the current time. Click to choose a different time in the drop-down list.
Daylight saving	Determines whether daylight saving is taken into account or not.
Azimuth	Displays the horizontal angle of the sun measured clockwise from the north. This setting is calculated automatically from the *Date*, *Time* and *Daylight saving* settings.
Altitude	Displays the vertical angle of the sun, measured from the horizon. This setting is calculated automatically from the *Date*, *Time* and *Daylight saving* settings.
Source vector	Displays the direction of the sunlight. This setting is calculated automatically from the *Date*, *Time*, *Daylight saving* and Geographic Location settings.
Rendered Shadow Details	
Type	Sets the shadow type. The options are: Sharp, Soft (mapped) and Soft (area).
Map size	This option applies to the Soft (mapped) shadow type. Select the desired value from the drop down-list.
Softness	Defines the appearance of the shadow edges.

Related Commands

Light - creates a new light.
Geographiclocation - sets the geographic location of the drawing.
Lightlist - opens the Drawing Explorer - Lights dialog.
Render - calculates a photorealistic image of the 3D model.

SupportFolder

Opens the main Support folder in a File dialog box.

Accessing the Command

command bar: *supportfolder*

: ***supportfolder***
Displays a file dialog with box the support folder open.

The default Support folder path is:
C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support
Support folder paths are saved through the SRCHPATH system variable.

Command Options

No Options

Related Commands

TemplateFolder - opens the Template folder in a File dialog box.

SvgOptions

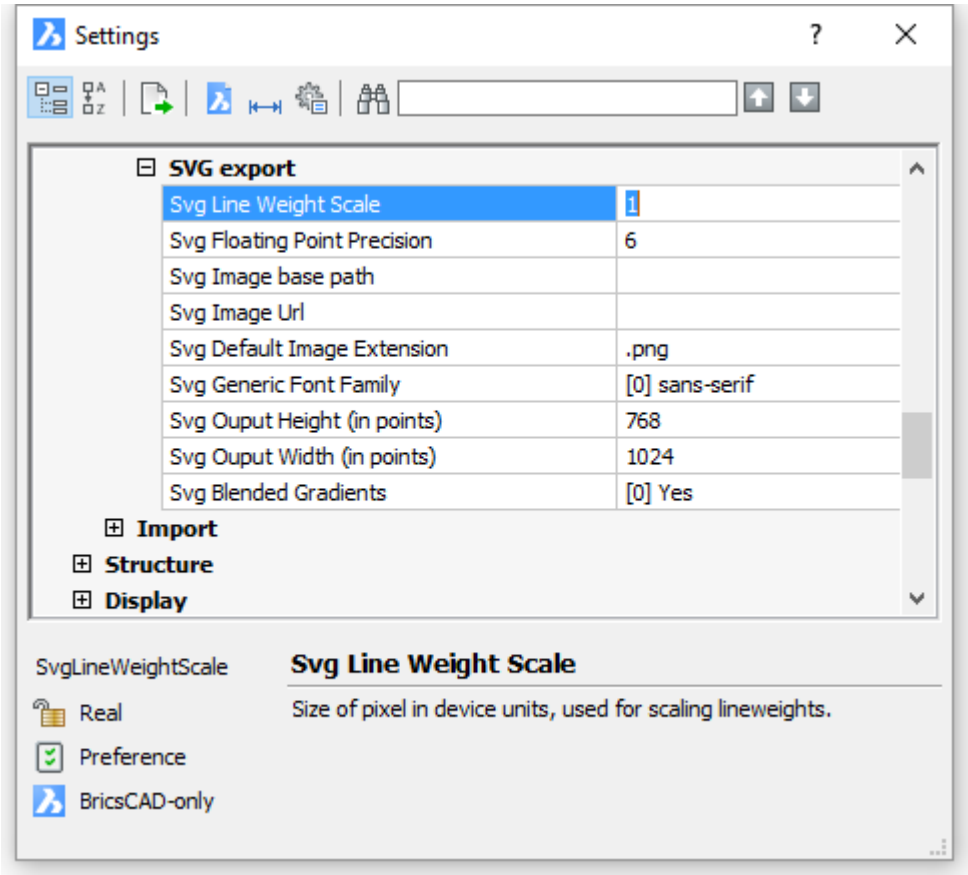
Displays the SVG (Scalable Vector Graphics) export options in the Settings dialog box.

Accessing the Command

command bar: *svgoptions*

: ***svgoptions***

Displays a dialog box:



Command Options

No options

Related Commands

Export - exports drawings in a variety of vector and raster formats.

Sweep

Creates a 3D solid by sweeping a closed 2D entity along a path.

Accessing the Command

command bar: *sweep*
menu bar: *Model| 3D Solids | Sweep*
toolbar: *3D Solids|*

: ***sweep***
Prompts you in the command bar:

Select profile entities: (Select one or more closed 2D entities*)

Entities in set: 1
Select path entities: (Select more closed 2D entities*, or right click to stop selecting 2D entities)

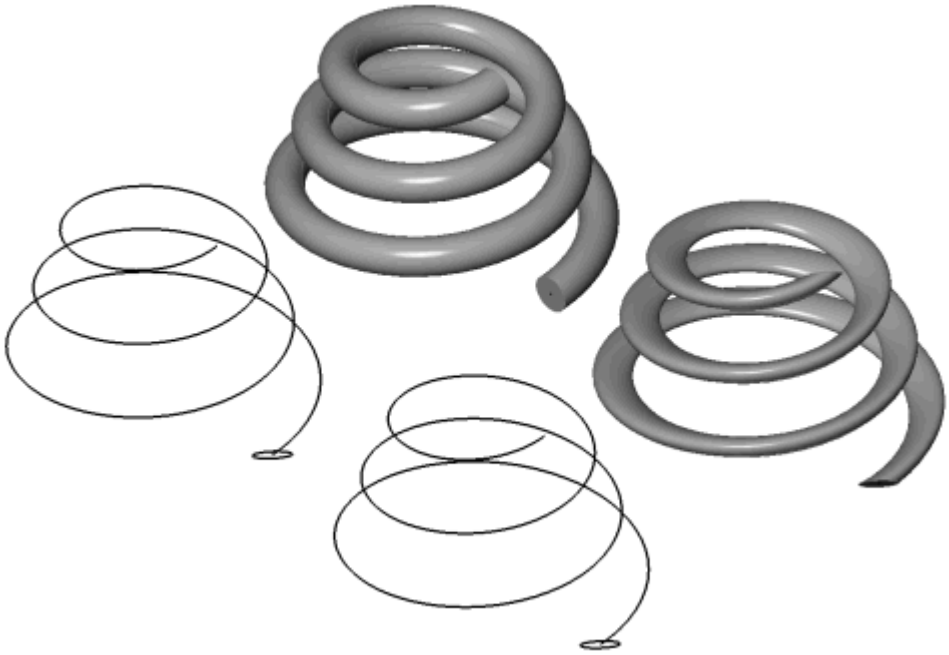
Select sweep path or [Alignment/Base point/Twist]: (Select a linear 2D entity or choose an option)

The selected closed 2D entities are swept along the path.

Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities.

* Detected boundaries are accepted.

Command Options

Option	Description
Alignment	By default 2D entities are placed perpendicular to the sweep path automatically. The Alignment option allows to sweep 2D entities at their current orientation. Prompts you: Align sweep entity perpendicular to path before sweep? [Yes/No]<Yes>: • Yes: The sweep entity is aligned perpendicularly • No: The sweep entity is not rotated. 

	In the images above a circle is swept along a helix path. Left: Alignment is ON (circle is placed perpendicularly to the sweep path) Right: Alignment is OFF (circle is not rotated)
Base point	Defines the point on the sweep entity that follows the path. Prompts you: Specify base point: pick a point in the drawing.
Twist	Specifies a twist angle for the swept entity or entities. The twist angle is the amount of rotation along the entire length of the sweep path. Prompts you: Enter twist angle or [allow Banking for a non-planar sweep path (Bank)] <0>: Do one of the following: • Type a value a the command prompt. • Select the *Bank* option: determines whether the swept entities will bank (rotate) naturally along a 3D sweep path (3D polyline, spline, or helix).

Grips Editing

Sweep solids can only be moved using grips.

Related Commands

dmExtrude - creates a 3D solids by extruding closed 2D entities or regions.
dmRevolve - creates 3D solids by revolution of closed 2D entities or regions about an axis.
dmThicken - creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids and wire entities such as lines, arcs, polylines, ...
Extrude - extrudes 2D entities to 3D solids.
Loft - creates 3D solids passing through two or more cross sections.
Revolve - revolves 2D closed entities to turn them into 3D solids.
RevSurf - creates a 3D mesh surface by revolving a linear entity around a line.

SysWindows

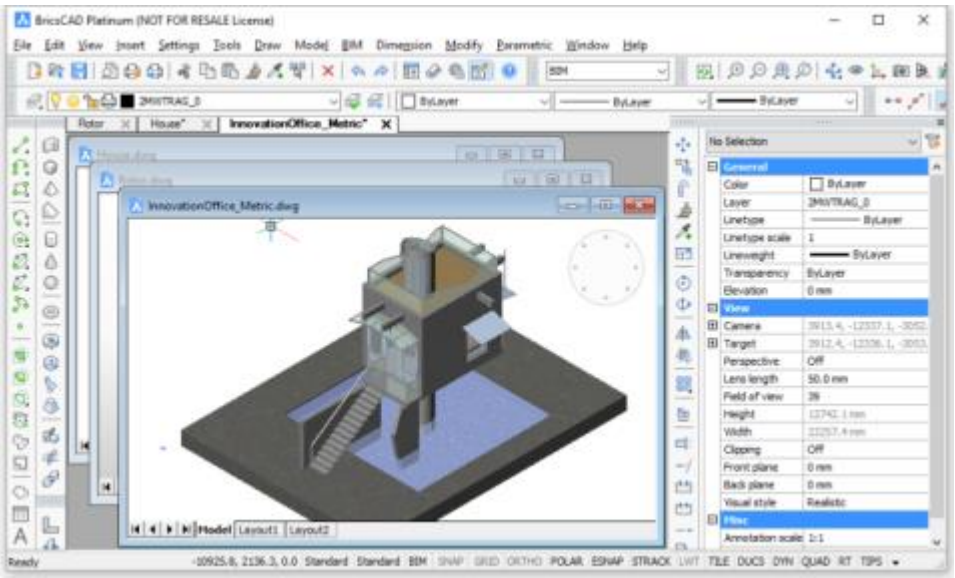
Arranges windows (short for "system windows").

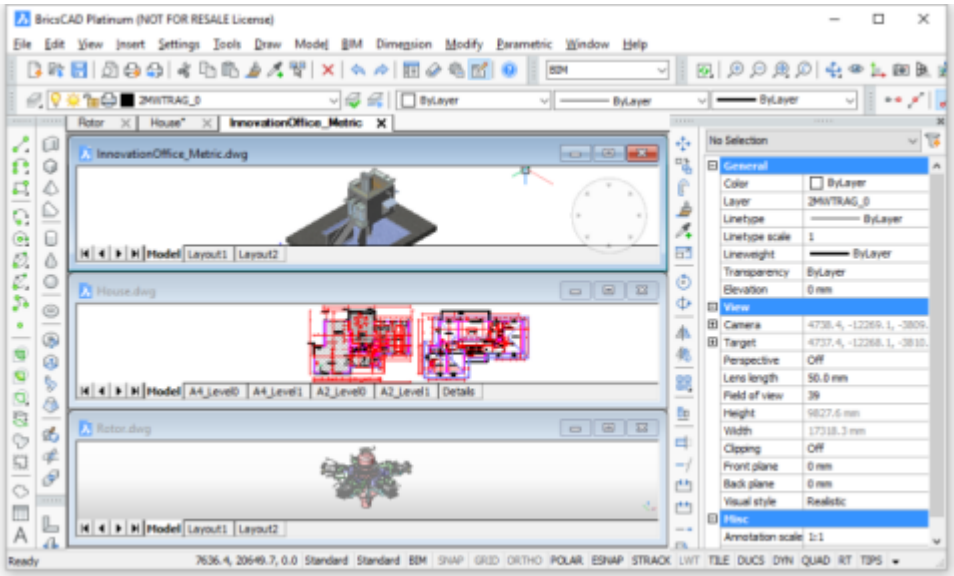
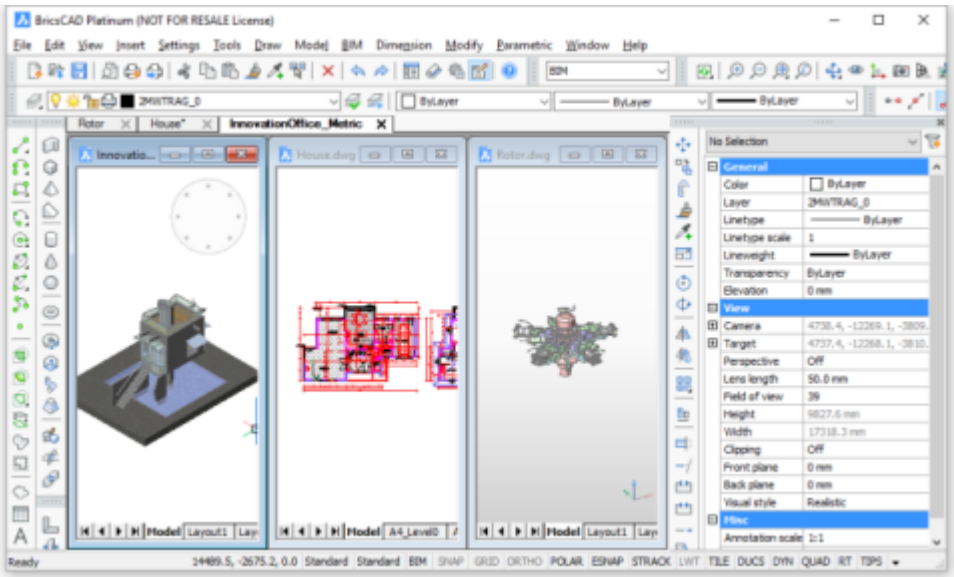
Accessing the Command

command bar: *syswindows*
menu bar: *Window | Cascade* or *Tile Vertically* or *Tile Horizontally*

: ***syswindows***
Prompts you in the command bar:
Arrange MDI windows [Cascade/tile Horizontally/tile Vertically/Arrange icons]
<Cascade>: *(Enter an option.)*

Command Options

Option	Description
Cascade	Overlaps windows (WCascade command): 

Horizontally	Arranges windows horizontally (WhTile command): 
Vertically	Arranges windows vertically (WvTile command): 
Arrange icons	Arranges icons of minimized windows at the bottom of the graphical screen (WiArrange command).

Related Commands

WCascade - cascades the windows.
WClose - closes the current window.
WCloseAll - closes all windows.
WhTile - tiles windows horizontally.
WiArrange - arranges iconized windows.
WvTile - tiles windows vertically.

Sheet Metal

SmAssemblyExport

Traverses the Assembly Structure of the current drawing, searches for 3D solids in its component, and applies the SmConvert and SmExport2d commands. The resulting .dxf files with unfolding information are saved in a user defined output folder and sorted by thickness. Parts which were processed with command warnings or errors are put to special folders. A HTML report listing all the solids in the assembly is generated, indicating their status and links to .dwg and .dxf files.
Each solid is classified:

- ”Sheet Metal” &endash; solid is a sheet metal part;
- ”Poor Sheet Metal” &endash; solid looks like a sheet metal design, user assistance and rework are needed;
- ”Not Sheet Metal” &endash; solid is not recognized as sheet metal design.

The command allows to quickly sort out non-sheet metal parts, therefore it can be applied to mixed assemblies containing both sheet metal and non-sheet metal parts.
The command can be applied to simple drawings with solids, to assemblies created in BricsCAD and to drawings imported by Communicator. It is recommended to set the ImportProductStructure Communicator user preference to 2 (= As mechanical components), else mechanical components will be exploded to solids, which increases the number of entities being processed by SmAssemblyExport, resulting in an increasing processing time and multiple repeats in the output.
In order to improve performance the Visual Style of the active viewport is best set to *2dWireframe*..

Accessing the Command

command bar: *smassemblyexport*
menu bar: *Sheet Metal | Unfold | Export Assembly to DXF*
toolbar: *Sheet Metal* |

ribbon: *Sheet Metal | Flatten* |

: ***commandline***
Prompts you in the command bar:
Select solids, component instances or [Settings] <Entire model>: (Select the to be processed or press Enter to include all the entities in assembly. Note that a component will be skipped only if it is not referred by any instance.)

Unless defined differently in the command settings, the results are saved in the folder of the processed drawing.

Command Options

Option	Description
Settings	Allows to set the output folder, assign a bend tabel or specify the DXF version; prompts you: Select setting [set Output folder/set bend Table/Dxf version/Back] <Back> - select an option. • Set Output Folder: Displays a File dialog to set the output folder. • Set Bend Table: Sets the bend table to be used during SmExport2d calls in the output process. Displays a File dialog to select the bend table file. • DXF version: Specifies the DXF version; prompts you: Select DXF version [R11/R12/R13/LT2/R14/LT95/2000/2004/2007/2010/2013/2018] <2013> - choose a DXF version or press Enter to accept the default. • Back: Returns to the previous prompt.

Related Commands

SmConvert - automatically recognizes flanges and bends in a 3D solid.
SmExport2d - exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

SmBendCreate

Converts hard edges (sharp edges between flange faces) into bends.

Accessing the Command

command bar: *smbendcreate*
menu bar: *Sheet Metal | Bend | Create Bend*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smbendcreate***
Prompts you in the command bar:
Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>: (Select a face or a hard edge or 3D solid or press Enter to include all sheet metal parts)
Entities in set: 1
Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>: (Select another face or hard edge or 3D solid or press Enter)*
* All selected items must be of the same kind: faces or hard edges.

Command Options

Option	Description
Flange faces	Creates bends between this flange and all adjacent flanges.
Hard edges	Creates bends on the selected hard edges.

Procedures

Converting hard edges to bends
Converting hard edges to junctions

Related Commands

SmConvert - automatically recognizes flanges and bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected faces.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge .

SmBendSwitch

Converts bends to lofted bends.

Accessing the Command

command bar: *smbendswitch*
menu bar: *Sheet Metal | Bend | Switch Bend*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smbendswitch***
Prompts you in the command bar:

Select bend faces/<Entire model>: (Select a bend or press Enter to select all bends in the model.)
Entities in set: 1

Select bend faces/<Entire model>: (Select a bend or press Enter to convert the selected bends.)

If FEATURECOLORS = ON the converted bends display in the SMCOLORLOFTEDBEND color.

Command Options

No options.

Related Commands

SmBendCreate - converts hard edges (sharp edges between flange faces) into bends
SmLoft - creates sheet metal part with lofted bends and flanges from two non-coplanar curves.
SmRepair - restores the 3D solid model of a sheet metal part by thickening one of its sides: all thickness faces become perpendicular to flange faces. Fixes problems specific to parts with lofted bend features: merges adjacent bends and provides tangential connection with flanges.

SmConvert

Automatically recognizes flanges, bends and lofted bends for selected 3D solids.

Accessing the Command

command bar: *smconvert*
menu bar: *Sheet Metal | Convert to Sheet Metal*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Create |*

: ***smconvert***
Prompts you in the command bar:
Select 3D solids or [Options/Entire model] <Entire model>: (Select a 3D solid or press Enter to include the entire model.)

Entities in set: 1

Select 3D solids or [Options/Entire model] <Entire model>: (Select more 3D solids or right click.)

Command Options

Option	Description
Options	By default the command tries to recognize flanges and bends, since they are easier to manufacture; prompts you: Select option [Prefer flanges and bends to form features/Back] <Back> - choose an option. • Prefer flanges and bends to form features: Allows to set the command behavior when the geometry can be represented as a sequence of flanges and bends or single form features. • Back: Returns to the previous prompt.
Entire Model	All solids in the entire model are analyzed and, if possible, converted to Sheet Metal bodies. The command recognizes sheet metal features (such as flanges and bends) in the input geometry. The body thickness of the sheet metal drawing is the criterium in case multiple solids are selected. Therefore selected bodies must have the same thickness, else an error message displays in the command window: "Could not convert the 3D solid to sheet metal. Recognized thickness is not equal the thickness of document: 5.000000 != 7.000000."

Related Commands

SmAssemblyExport - analyzes the assembly structure of the current drawing, executes the SmConvert and SmExport2d commands, and saves the resulting .dxf files with unfolding information in an output folder.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted.
SmDissolve - removes sheet metal data from the selected faces.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmForm - converts a selected set of faces to a form feature.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.
SmRethicken - restores the 3D solid model of a sheet metal part by thickening one of its sides (all thickness faces become perpendicular to flange faces).

SmDelete

Removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.

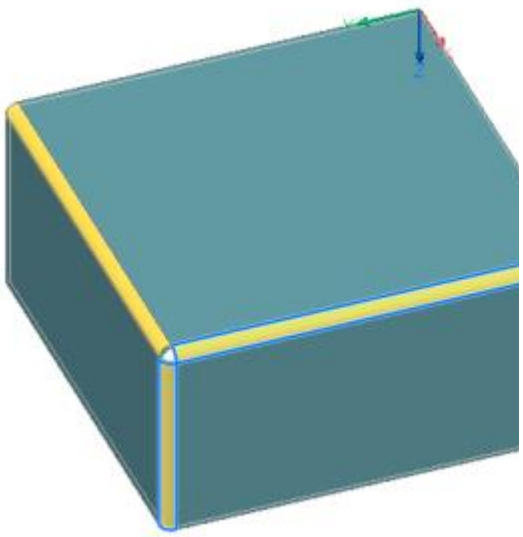
Accessing the Command

command bar: *smdelete*
menu bar: *Sheet Metal | Delete Feature*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

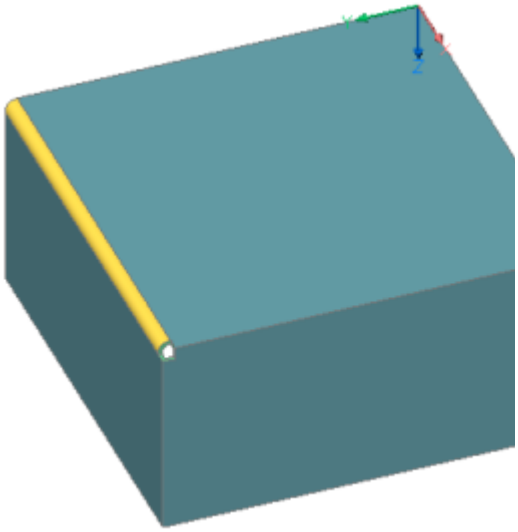
: ***smdelete***
Prompts you in the command bar:

Select feature faces or thickness faces: (Select an item.)
Entities in set: 1

Select feature faces or thickness faces: (Select another item or right click.)



Selected bend faces



The selected bends are replaced by hard edges

Command Options

No options.

Related Commands

SmBendCreate - converts hard edges (sharp edges between flange faces) and junctions into bends.
SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDissolve - removes sheet metal data from the selected faces or entities.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlangeSplit - splits a flange along a polyline drawn on its face.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge .

SmDissolve

Removes sheet metal data from the selected features.
The command can dissolve sheet metal features on 3D solid faces. Dissolved features are no longer listed in the Mechanical Browser. Faces of dissolved features can be moved, rotated or push-pulled freely.

Accessing the Command

command bar: *smdissolve*
menu bar: *Sheet Metal | Dissolve*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smdissolve***
Prompts you in the command bar:

Select faces on models or [Entire model] <Entire model>: Select a face, or entities, or right click to select the entire model.
Entities in set: 1

Select faces on models: (Select a face or right click to stop.)

Command Options

Option	Description
Entire model	Selects all faces in the entire model.

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmForm - converts a selected set of faces to a form feature.
SmLoft - creates sheet metal part with lofted bends and flanges from two non-coplanar curves.

SmExport2d

Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

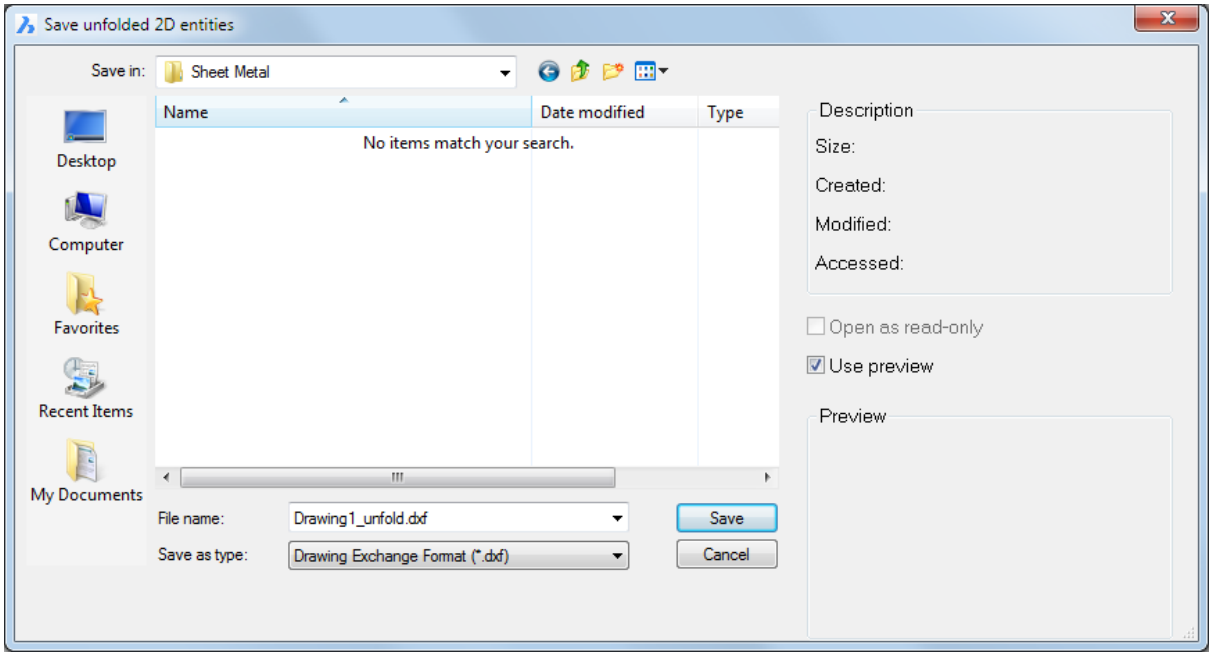
Accessing the Command

command bar: *smexport2d*
menu bar: *Sheet Metal | Unfold | Export to DXF*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Flatten |*

: ***smexport2d***
Prompts you in the command bar:

Select a flange face to start exporting: (Identify the start face.)

Displays a file dialog box:



Select the file format: dwg or dxf

Command Options

No options.

Related Commands

SmAssemblyExport - analyzes the assembly structure of the current drawing, executes the SmConvert and SmExport2d commands, and saves the resulting .dxf files with unfolding information in an output folder.
SmExportOSM - exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system).
SmUnfold - creates an unfolded 2D or 3D representation of a sheet metal part.

SmExportOSM

Exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system).

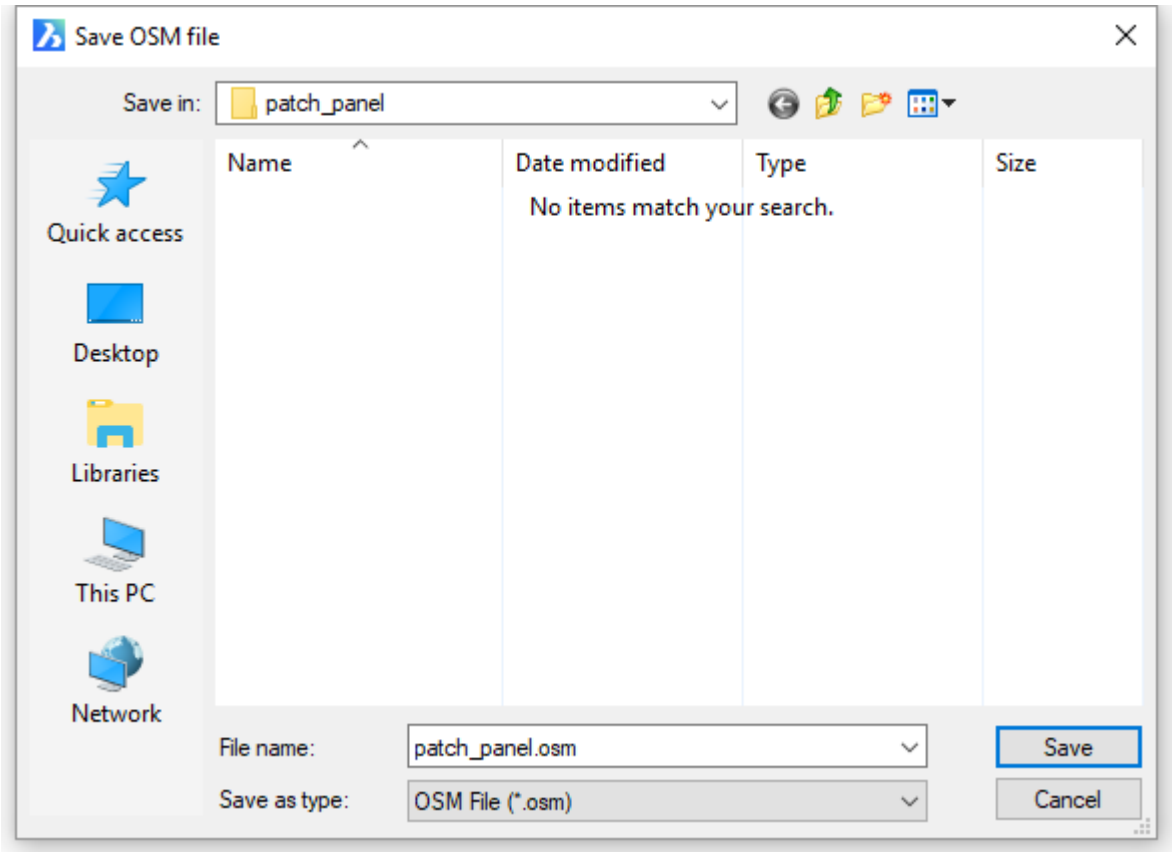
Accessing the Command

command bar: *smexportosm*
menu bar: *Sheet Metal | Unfold | Export to OSM*
toolbar: *Sheet Metal |* OSM
ribbon: *Sheet Metal | Flatten |* OSM

: ***smexportosm***
Prompts you in the command bar:

Select a sheet metal body or [set the Accuracy of the approximation/minimal edge Length]: (Select a sheet metal body or choose an option.)

Displays a file dialog:



Command Options

Option	Description
Set the accuracy of the approximation	The OSM file format requires splines to be replaced by linear segments and circular arcs. The accuracy of this approximation (allowable deviation of a segment/arc from the spline) can be specified for each run of the SmExportOSM command. The .osm part is saved as mm- or inch-based, depending on the value of the document MEASUREMENT system variable. Document INSUNITS are taken into account for proper scaling.
Minimal Edge Length	Controls the minimal length of edges in the resulting .osm file.

Related Commands

SmAssemblyExport - traverses the Assembly Structure of the current drawing, searches for 3D solids in its component, and applies the SmConvert and SmExport2d commands.
SmExport2d - exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
SmUnfold - creates an unfolded 2D or 3D representation of a sheet metal part.

SmExtrude

Creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.

Accessing the Command

command bar: *smextrude*
menu bar: Sheet Metal | Bend | Create Bend
toolbar: *Sheet Metal* |
ribbon: *Sheet Metal* | *Create* |

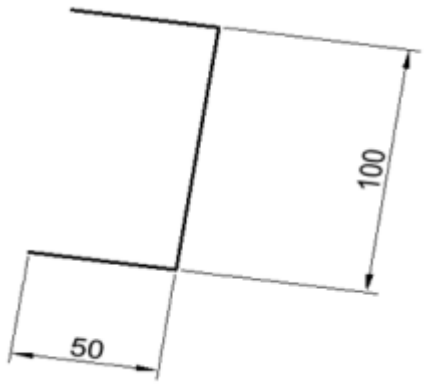
: ***commandline***
Prompts you in the command bar:

Select a polyline: (Select a polyline.)

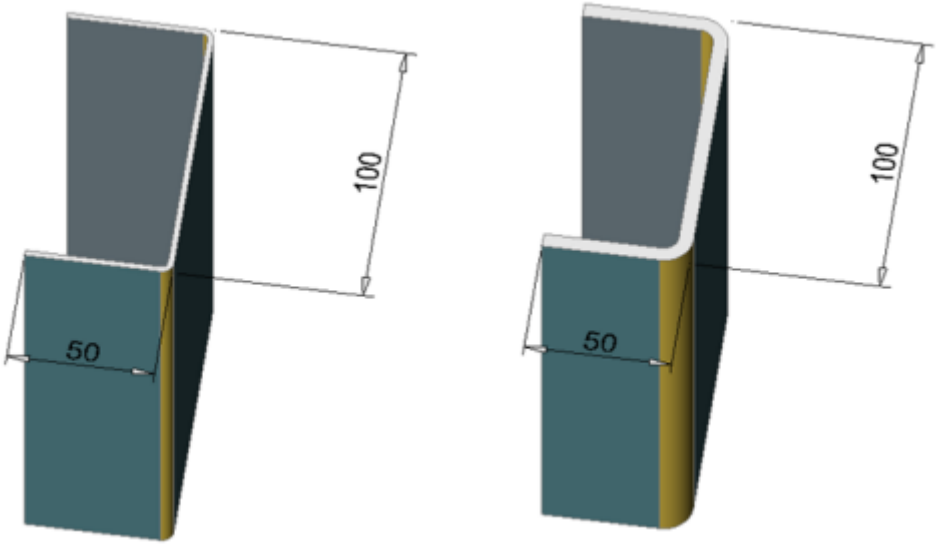
The polyline is extruded dynamically.

Specify height of extrusion: (Pick a point or type a value.)

The thickening direction of the extruded surface is applied for each flange separately, in order to preserve the overall dimensions of polyline:



Selected Polyline



The overall dimensions of the Sheet Metal part match the dimensions of the extruded polyline.

Command Options

No command options.

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmFlangeBend - allows to bend an existing flange along a line, obeying k-factor for given bend radius.
SmFlip - changes reference side for selected flanges with optional shift of flange geometry by thickness.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.

SmFlangeBase

Creates a base (initial) flange of a sheet metal part from a closed planar profile.

Accessing the Command

command bar: *smflangebase*
menu bar: *Sheet Metal | Create Base Flange*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Create |*

: ***smflangebase***
Prompts you in the command bar:
Select closed polyline or region: (Select a closed polylines or region)

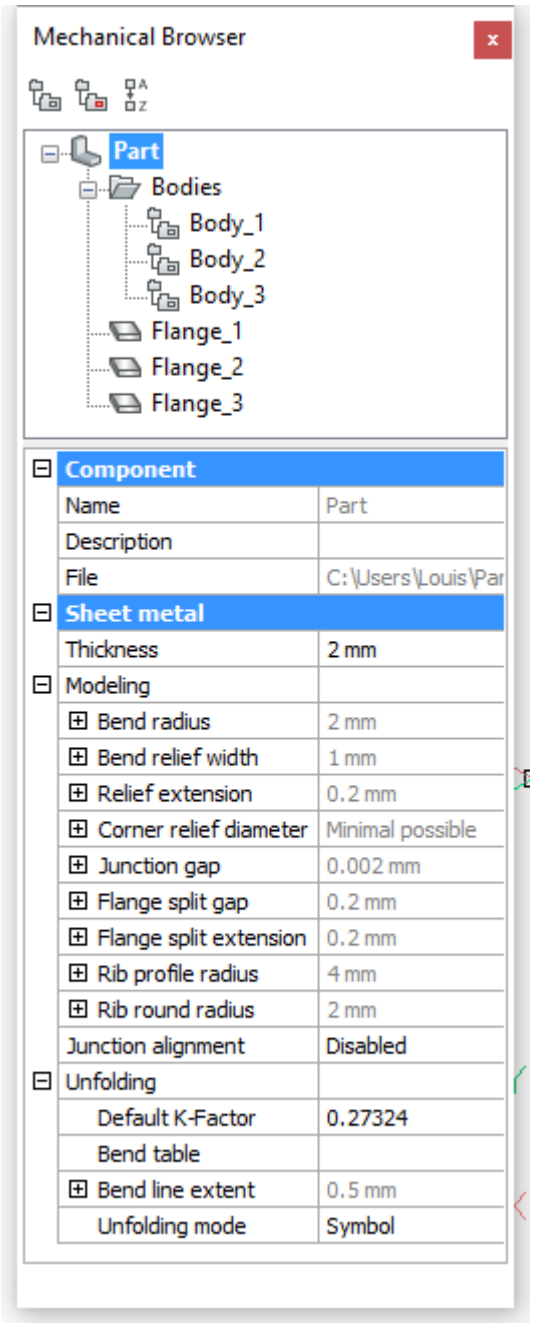
Entities/subentities in set: 1

Select closed polyline or region: (Select another closed polylines or region or right click to confirm)

Make flange base Up/Middle/Down/<Accept model>: (Select an option or press the Ctrl-key to cycle the various options; press Enter or right click to accept the selected option.)

A base flange is created for each selected closed polyline or region.

In the Mechanical Browser a body and a flange is added for each entity:



Command Options

Option	Description
Up	The flange is extruded on top of the base entity.
Center	The flange is extruded on both sides of the base entity.
Down	The flange is extruded below the base entity.

Procedures

Changing the flange thickness

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDissolve - removes sheet metal data from the selected faces or entities.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlip - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange.
SmLoft - creates sheet metal part with lofted bends and flanges from two non-coplanar curves.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.
SmRepair - restores the 3D solid model of a sheet metal part by thickening one of its sides (all thickness faces become perpendicular to flange faces).

SmFlangeBend

Allows to bend an existing flange along a line, obeying k-factor for given bend radius.

Accessing the Command

command bar: *smflangebend*
menu bar: *Sheet Metal | Bend | Flange Bend*
toolbar: *Sheet Metal | Bend |*
ribbon: *Sheet Metal | Modify | Bend |*
Quad: *Sheet Metal |*

: ***smflangebend***
Prompts you in the command bar:
Select a flange face: (Select a major face of flange.)

Select line or edge to bend the solid or draw a [New line] <New line>: (Select a line or an edge.)

Pick a point on the side that will be moved: (Pick a point on the side of the line that will be moved.)

Position the flange [Angle/Radius/Switch side/ACcept] <Accept>: (Choose an option or press Enter.)

Command Options

Option	Description
New line	Allows to draw a line to define the bend location; prompts you: Start point of the line - Pick a point. End point of the line - Pick a point.
Angle	Allows to set the bend angle; prompts you: **Enter bend angle [Back] <Back>:** Do one of the following: • Type a value. • Press Enter to go back without changing the bend angle.
Radius	The default bend radius is defined by the *Modeling/Bend Radius* property of the sheet metal part. Allows to define the bend radius; prompts you: **Enter bend radius [Back] <Back>:** Do one of the following: • Type a value. • Press Enter to go back without changing the bend radius.
Switch side	Allows to move the part of the flange on the other side of the line or edge.
Accept	Creates the bend at the current angle and radius.

Procedures

To bend multiple flanges simultaneously

Related Commands

SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlangeRotate - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent

SmFlangeConnect

Closes gaps between two arbitrarily oriented flanges.

Accessing the Command

command bar: *smflangeconnect*
menu bar: *Sheet Metal | Connect Flanges*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smflangeconnect***
Prompts you in the command bar:

Select planar thickness faces of two flanges: (Select the face on the first flange.)

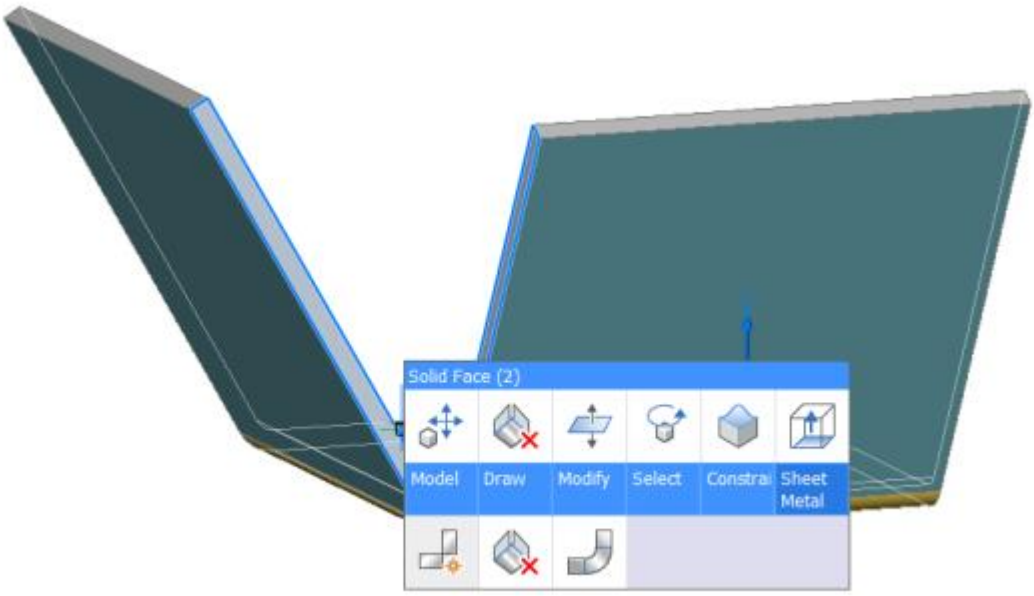
Entities/subentities in set: 1

Select planar thickness faces of two flanges: (Select the face on the first flange.)
The flanges are connected.

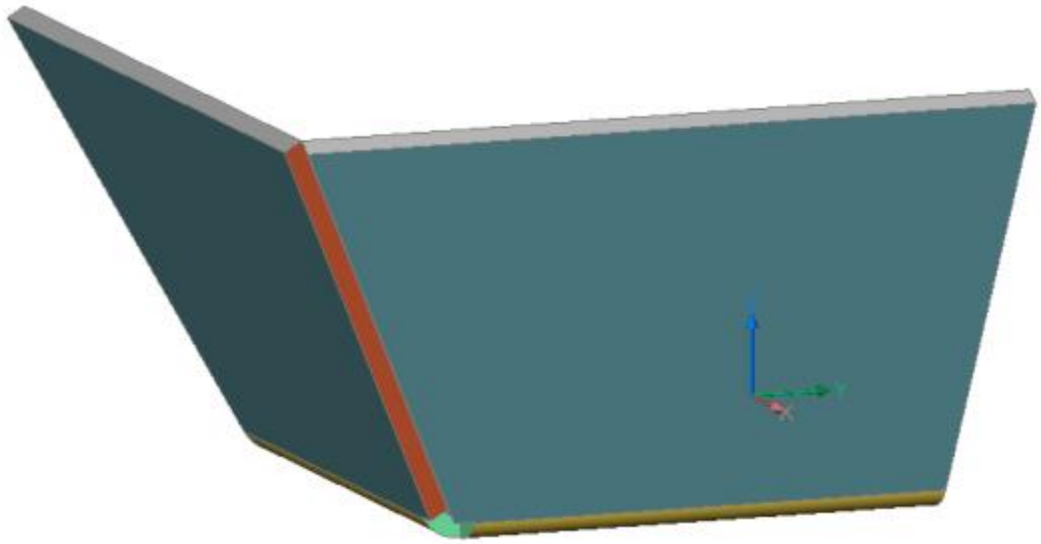
If SELECTIONPREVIEW is 2 or 3, thickness faces of flanges highlight. Press the TAB key to select obscured faces.

Quad cursor menu

Select the planar thickness faces of two flanges, then choose *Connect Flanges* ( ) in the *Sheet Metal* command group.



Select the thickness faces



A junction is created

Command Options

No command options.

Procedures

Connecting flanges.

Related Commands

SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlangeRotate - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.
SmFlangeSplit - splits a flange along a line drawn on its face.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

SmFlangeEdge

Creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.

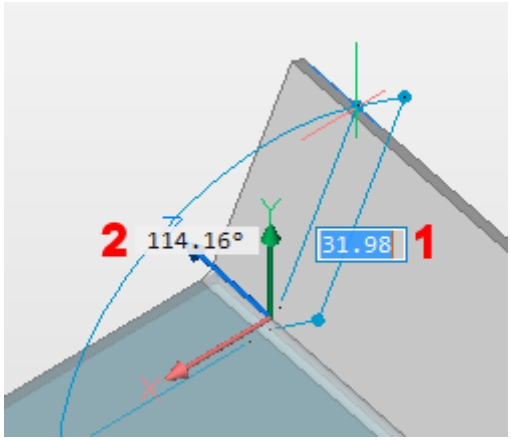
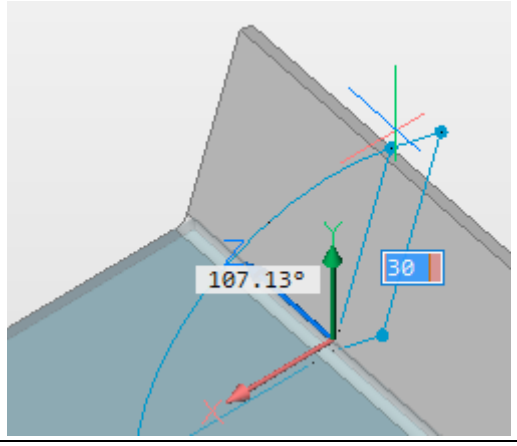
Accessing the Command

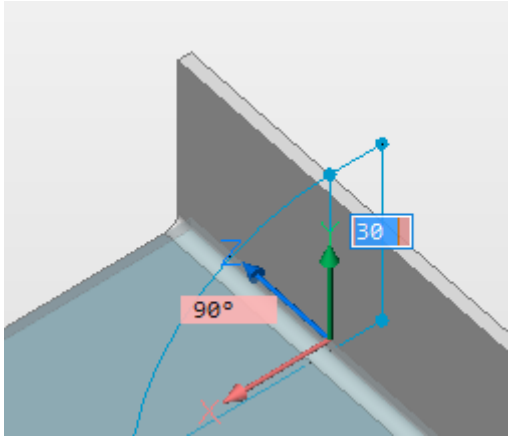
command bar: *smflangeedge*
menu bar: *Sheet Metal | Create Edge Flange*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smflangeedge***
Prompts you in the command bar:

Select edges on flanges: (Select edges, then right click to proceed.)
Entities/subentities in set: 1
Position the end of the flange [Angle/Length/Taper angle/Width]: (Specify a point in 3D space or type a value in the dynamic height and angle fields)

Command Options

Option	Description
Dynamic dimensions	Define the height and angle of the flange by typing values in the dynamic height (1) and angle (2) entry fields:  Enter a value in the height field, then press the TAB key to jump to the angle field. 

	Type a value in the angle field, then do one of the following: • Press Enter twice to accept the current values. • Press the TAB key to enter a different value in the height field, then press Enter to accept the current values. • Keep pressing the TAB key to switch between the height and angle field to enter a different value, then press Enter to accept the current values. 
Angle	Allows to first define the angle, then the height of the flange.
Length	Allows to first define the length, then the angle of the flange.
Taper Angle(*)	Allows to create a flange with one or two tapered side faces. Prompts you: **Enter first taper angle [Back/Skip] <Skip>:** Do one of the following: • Type a value in the taper angle entry field to define the taper angle for the first side face. • Select the *Skip* option to keep the first side face orthogonal. • Select the *Back* option to undo the current *Taper angle* procedure. Enter second taper angle [Back/Skip] <Skip>: Do one of the following: • Type a value in the taper angle entry field to define the taper angle for the second side face. • Select the *Skip* option to keep the second side face orthogonal. • Select the *Back* option to undo the current *Taper angle* procedure. **Position the end of the flange [Angle/Length/Taper angle/Width]:** Do one of the following:

	• Click a point or use the Dynamic Dimensions procedure to define the flange height and angle. • Choose the Width option to define a width different from the length of the selected edge. • Choose the *Taper angle* option again to define different taper angles.
Width(*)	By default the flange width equals the length of the selected edge. The Width option allows to define a different width. Prompts you: **Enter first offset [Back/Skip] <Skip>:** Do one of the following: • Type a value in the dynamic field to define the offset from the start point of the edge. • Select the *Skip* option to keep the first side edge of the flange at the start point of the edge. • Select the *Back* option to undo the current *Width* procedure. **Enter second offset [Back/Skip] <Skip>:** Do one of the following: • Type a value in the dynamic field to define the offset from the end point of the edge. • Select the *Skip* option to keep the second side edge of the flange at the end point of the edge. • Select the *Back* option to undo the current *Width* procedure. **Position the end of the flange [Angle/Length/Taper angle/Width]:** Do one of the following: • Click a point or use the Dynamic Dimensions procedure to define the flange height and angle. • Choose the Taper angle option to define tapered side faces • Choose the *Width* option again to define different offsets.

(*) The *Taper Angle* and *Width* options are not available if multiple edges are selected.

Related Commands

SmBendCreate - converts hard edges (sharp edges between flange faces) and junctions into bends.
SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected faces.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmFlangeBend - allows to bend an existing flange along a line, obeying k-factor for given bend radius.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmFlangeRotate - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.
SmFlangeSplit - splits a flange along a polyline drawn on its face.
SmFlip - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.
SmRepair - restores the 3D solid model of a sheet metal part by thickening one of its sides (all thickness faces become perpendicular to flange faces).

SmFlangeRotate

Rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.

Accessing the Command

command bar: *smflangerotate*
menu bar: *Sheet Metal | Rotate Flange*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Edit |*

: ***smflangerotate***
Prompts you in the command bar:

Select a flange face to rotate: (Select the face of a flange.)

A dynamic angle field displays the angle between the selected flange and the base flange.

Press the TAB key to display the absolute rotation angle.

(Type a value in the dynamic angle field and press Enter to confirm.)

Command Options

No options.

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmFlangeBend - allows to bend an existing flange along a line, obeying k-factor for given bend radius.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlip - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange.

SmFlangeSplit

Splits a flange along a polyline drawn on its face.

Accessing the Command

command bar: *smflangesplit*
menu bar: *Sheet Metal | Split | Split Flange* and *Use Smart Split Flange*
toolbar: *Sheet Metal |* and
ribbon: *Sheet Metal | Modify |* and

: ***smflangesplit***
Prompts you in the command bar:

Select a flange face: (Select the face of flange.)

Select lines, edges to split the flange or [use SMart split/draw a New line/Settings] <draw a New line>: (Select a polyline or edge on the flange or choose an option.)

Command Options

Option	Description
Use Smart Split	Allows to split the flange by selecting a single point at a flange corner. Variants for placing a second point will be automatically suggested; prompts you: Select point for split or [use split along a given Line/**Settings**] <use split along a given Line> - choose an option: • Select point for split: Snap to a vertex of the selected flange. Prompts you: Make split [Center/Left/Right/Accept] <Accept>: Repeatedly hit the Ctrl-key to cycle the options, then press Enter to accept the current split. If TIPS = ON, the currently selected option is indicated in the Tips widget at the bottom of the screen. • use split along a given Line: returns to the initial prompt: Select lines, edges to split the flange or [use SMart split/draw a New line/Settings] <draw a New line>:
New Line	Allows to draw a line on the selected flange. Prompts you: Start point of the line: Specify the start point of the line. End point of the line: Specify the end point of the line.

Settings	Prompts you: Settings [Convert bend to junction/Make orthogonal bend split/Heal coincident miter faces] - choose an option. • Convert bend to junction: Allows to recognize an L-corner on a flange and automatically convert part of the split bend into a junction; prompts you: Convert bend to junction [ON/OFF] <On> - choose ON or OFF or press Enter to accept the current setting. • Make orthogonal bend split: Determines whether a bend will be split orthogonally to the bend axis if the bend line crosses the bend at an angle; prompts you: Make orthogonal split with bends [ON/OFF] <On> - choose ON or OFF or press Enter to accept the current setting. • Heal coincident miter faces: Switches the mode for the recognition of miter-like design made of coincident faces; OFF = keep; ON = convert to regular miters; prompts you: Heal coincident miter faces [ON/OFF] <On> - choose ON or OFF or press Enter to accept the current setting.

Procedures

To split a flange

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

SmFlip

Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. Optionally shifts the flange over the thickness of the sheet metal part.

Accessing the Command

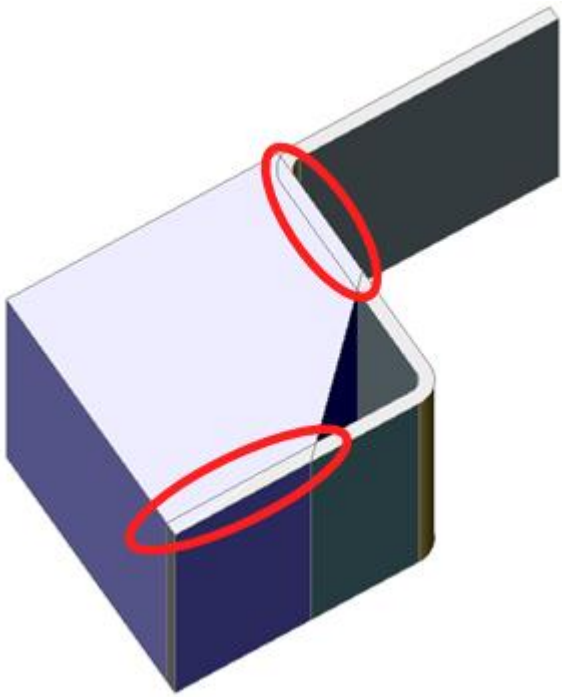
command bar: *smflip*
toolbar: *Flange* |
ribbon: *Sheet Metal* | *Modify* | *Flip Flange*

: ***smflip***
Prompts you in the command bar:
Select flange faces and allow rebuild geometry: [Disallow rebuild geometry/Entire model] <Entire model>: (Select a face or press Enter to flip sides for all flanges.)
Entities in set: 1
Select flange faces and allow rebuild geometry [Disallow rebuild geometry/Entire model]: (Select more faces or press Enter.)

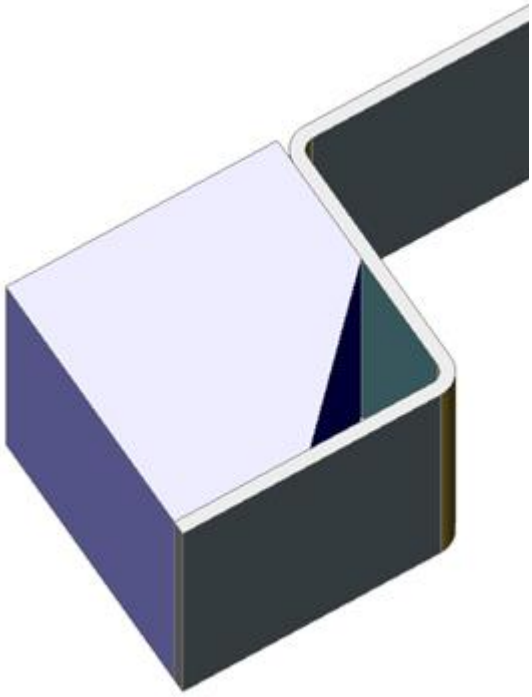
Command Options

Option	Description
Disallow rebuild geometry	Turns OFF the shift of flanges by thickness, in this mode reference sides for selected flanges are just swapped. The command prompt now reads: **Select flange faces and disallow rebuild geometry [Allow rebuild geometry/Entire model] <Entire model>:**
Allow rebuild geometry	Turns ON the shift of flanges by thickness. The command prompt now reads: **Select flange faces and allow rebuild geometry [Disallow rebuild geometry/Entire model]:**
Entire Model	Applies flip operation to all flanges in the drawing.

Changing reference faces with shift by thickness might be necessary if SmExtrude creates flanges which collide with another part:



Collisions between a solid and sheet metal flanges



Collisions solved after applying SmFlip.

Related Commands

SmConvert - automatically recognizes flanges and bends in a 3D solid.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed planar profile.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmParametrize - creates a consistent set of 3D constraints for a selected sheet metal part.

SmForm

Converts a selected set of faces to a form feature or explodes form feature into flanges and bends. In *Create* mode the command allows to define a form feature by selecting the faces manually, which can be necessary if automatic recognition by SmConvert does not produce the expected result. E.g.: if you insert and dissolve the *Cardguide* form feature from the library, SmConvert recognizes two forms features, because their geometry is separated. Manual selection of the form feature faces when executing SmForm solves the problem.
Form features created by SmForm have the same set of basic operations as if they were created by BmInsert or recognized by SmConvert.
In *Explode* mode the command dissolves the form feature and tries to recognize its geometry as flanges and edges. This can be helpful if SmConvert does not correctly recognize the design as a form feature.

Accessing the Command

command bar: *smform*

: ***smform***
Prompts you in the command bar:
Select form feature faces [Create new form feature/Explode form feature] <Create new form feature>: (Select faces or choose an option to switch the command mode to Create or Explode.)

Command Options

Option	Description
Create new form feature	Sets the command in *Create* mode, which creates new form features. Prompts you to select faces which do not belong to any existing feature.
Explode form feature	Sets the command in *Explode* mode, which explodes existing form features into flanges and bends. Prompts you to select faces of existing form features. At least one face of each form feature to be exploded must be selected.

Procedures

Working with form features

Related Commands

BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.
SmConvert - automatically recognizes flanges and bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected features.

SmImprint

Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.

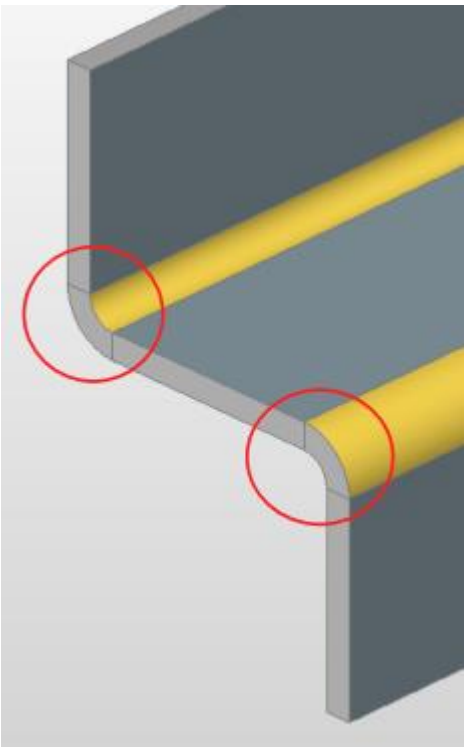
Accessing the Command

command bar: *smimprint*
menu bar: *Sheet Metal | Split thickness face*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smimprint***
Prompts you in the command bar:
Select a thickness face: (Select a thickness face)

Entities in set: 1

4 edge(s) were imprinted.



Command Options

No options.

Procedures

To split a thickness face.

Related Commands

SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmJunctionSwitch - allows changing a symmetrical junction feature to one with overlapping faces.

SmJunctionCreate

Converts hard edges (sharp edges between flange faces) and bends into junctions.

Accessing the Command

command bar: *smjunctioncreate*
menu bar: *Sheet Metal | Create Junction*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smjunctioncreate***
Prompts you in the command bar:
Select hard edges or bends or flanges or 3D solids [Entire model/SEttings] <Entire model>: (Select a flanges, hard edges, bends or solids, or choose an option.)

Entities in set: 1

Select hard edges or bends or flanges or 3D solids [Entire model/SEttings] <Entire model>: (Select more items or right click.)

Command Options

Option	Description
Entire model	Analyzes the entire model to detect hard edges.
Settings	Prompts you: Select settings [Heal coincident junction faces/Back] - choose an option. • Heal coincident junction faces: Switches the mode for recognition of junction-like coincident faces and conversion to regular junctions; prompts you: Heal coincident faces [ON/OFF/Back] <OFF> - choose an option. • Back: Returns to the previous prompt.

Procedures

Converting hard edges to junctions
Converting hard edges to bends

Related Commands

SmBendCreate - converts hard edges (sharp edges between flange faces) and junctions into bends.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected faces or entities.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmImprint - splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
SmJunctionSwitch - allows changing a symmetrical junction feature to one with overlapping faces.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

SmJunctionSwitch

Allows changing a symmetrical junction feature to one with overlapping faces.

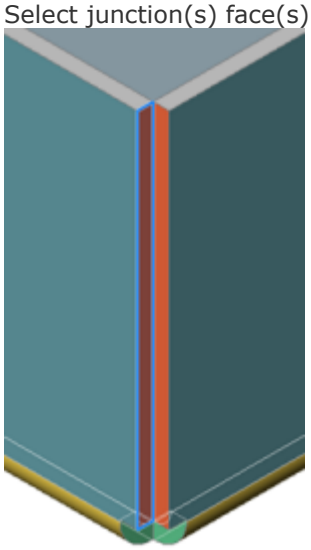
Accessing the Command

command bar: *smjunctionswitch*
menu bar: *Sheet Metal | Switch Junction type*
toolbar: *Sheet Metal |*
ribbon:*Sheet Metal | Modify |*

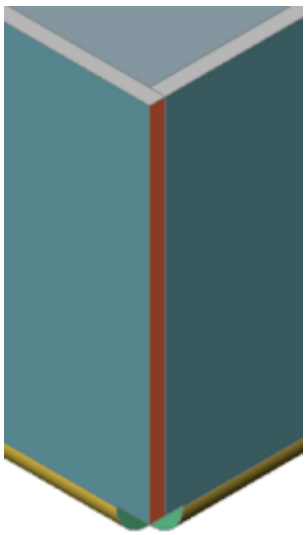
: ***smjunctionswitch***
Prompts you in the command bar:
Select junction(s) face(s): (Select the junction face of the first flange)

Entities in set: 1

Select junction(s) face(s): (Select the junction face of the second flange, then right click.)



The junction face of the first flange overlaps the junction face of the second flange.



Command Options

No command options.

Procedures

Changing the junction type

Related Commands

SmImprint - splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.

SmLoft

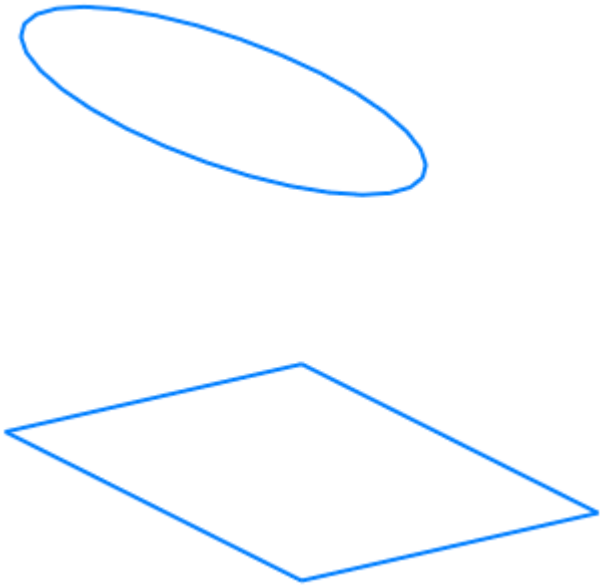
Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.

Accessing the Command

command bar: *smloft*
menu bar: *Sheet Metal | Create Sheet Metal Loft*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Create | Lofted Bend*

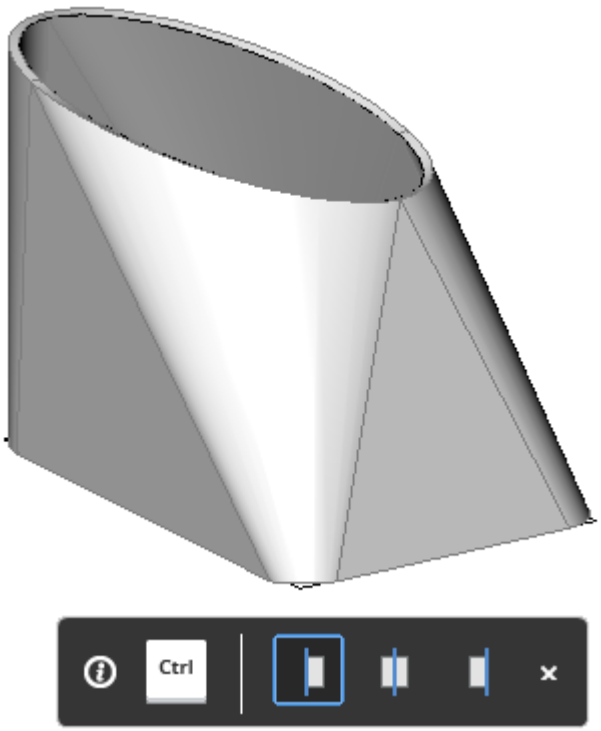
: ***smloft***
Prompts you in the command bar:

Select two cross sections: (Select two, non coplanar open or closed 2D entities.)

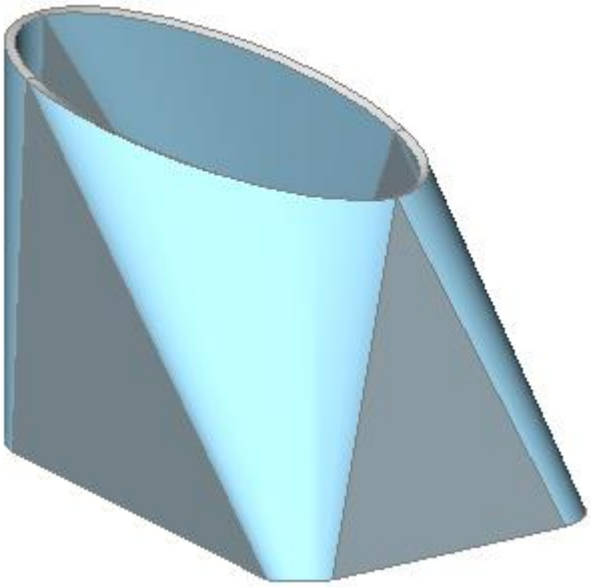


Set [fillets Radius/Thickness/Single bend/Flanges with bends] <Flanges with bends>: (Press Enter or choose an option.)

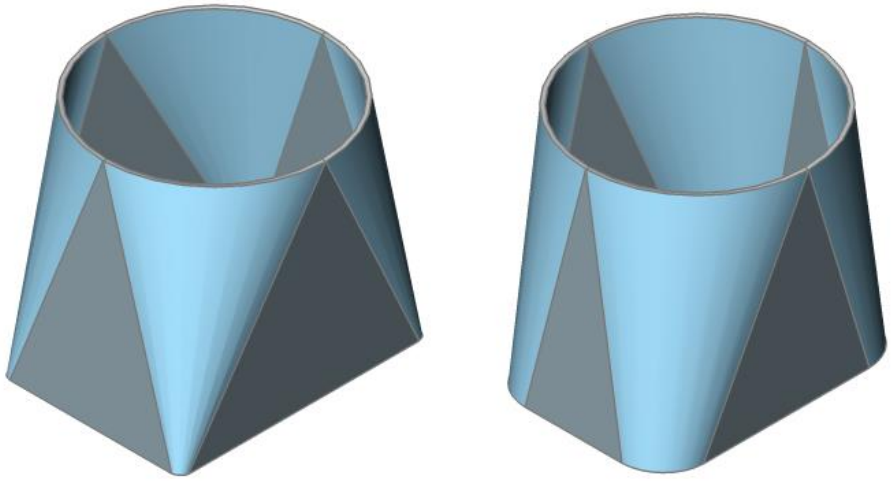
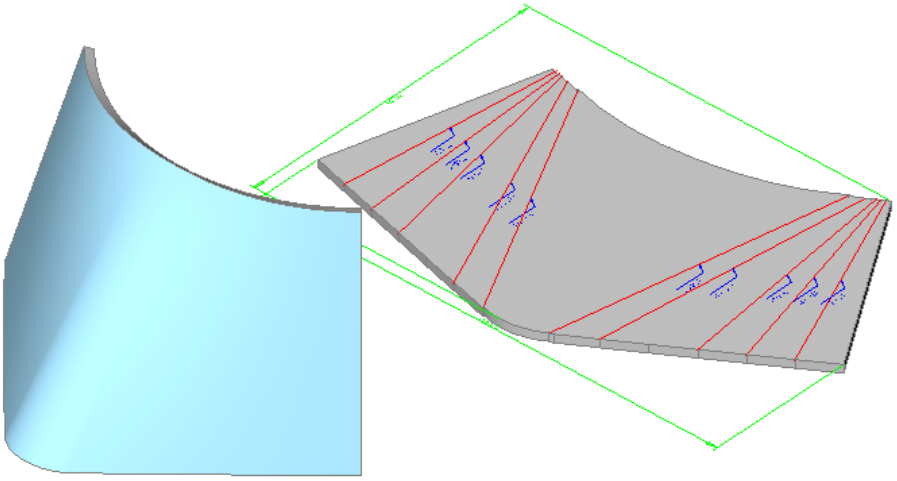
Set [fillets Radius/Thickness/Single bend/Flanges with bends/thicken Inside/Both sides/thicken Outside] <thicken Inside>: (Press Enter or choose an option.)

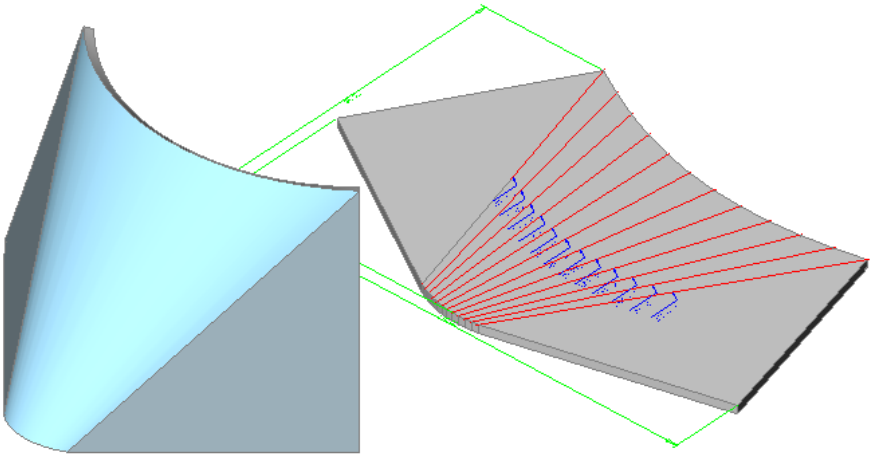
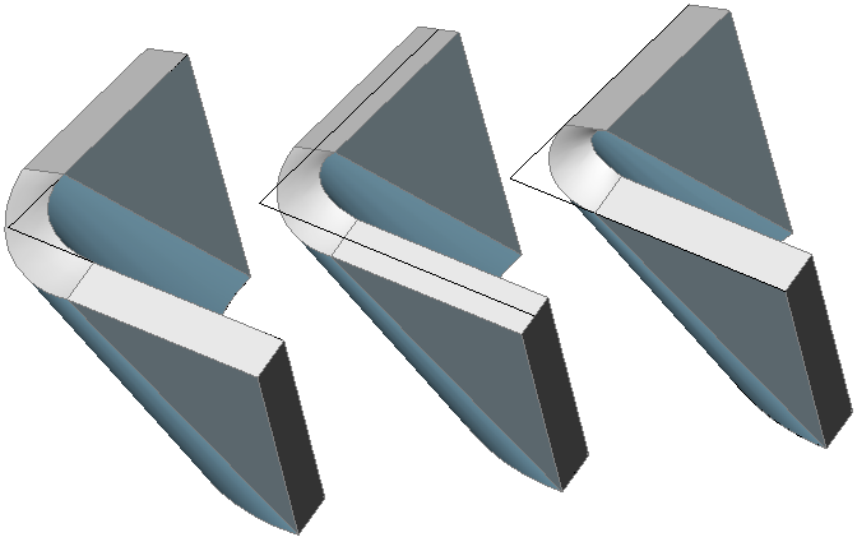


Hit the Ctrl-key to adjust the alignment: Outside, Both Sides or Inside



Command Options

Option	Description
Set Fillets Radius	Sets radius of fillets which will be created on straight lines connection. Prompts you: **Select fillets radius <auto>:** Do one of the following: • Type a value for the fillet radius. • Press Enter to automatically calculate the minimal value for the fillet radius, which allows to create the model with the current thickness.  *Automatically calculated radius (left); user defined radius (right)*
Thickness	Sets model thickness. Prompts you: Select thickness <2.000000>: Type a value or press enter to accept the default value.
Single Bend	Creates only lofted bend feature without any flanges.  *Single lofted bend and its unfolded view.*

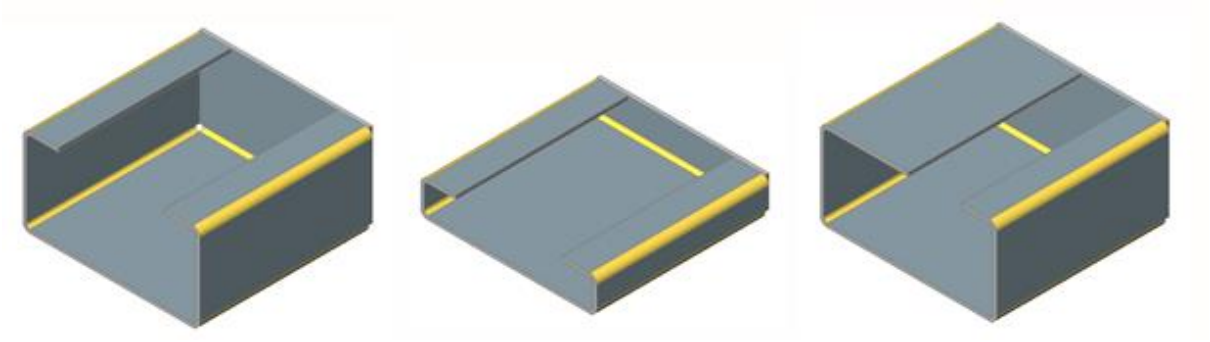
Flanges with Bends	Creates a flange feature for each straight segment of the selected 2D entities or a lofted bend feature for curved segments.  *Lofted bend with flanges and its unfolded view.*
	 *From left to right: thickening outside, both sides an inside.*
Inside	Thickens at the outside of the selected 2D entities.
Both Sides	Thickens at both sides of the selected 2D entities.
Outside	Thickens at the inside of the selected 2D entities.

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed 2D entity.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.

SmParametrize

Creates a consistent set of 3D constraints for a selected sheet metal part.
The command takes into account implicit constraints from the sheet metal features and ensures that the constraint system is not overdefined. Editing the parameter allows to obtain a series of similar sheet metal parts with different dimensions. Please notice that for the best results, the sheet metal part should have a complete set of features, including flanges, bends, junctions and bend and corner reliefs.
Example of a fully parametrized sheet metal part (left) and different sets of parameters applied (center and right):



Accessing the Command

command bar: *smparametrize*
menu bar: *Sheet Metal | Parametrize*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | 3D Constraints |*

: ***smparametrize***
Prompts you in the command bar:
Select 3D solids to parametrize or [Entire model/SEttings] <Entire model>: (Select a sheet metal part or press Enter to parametrize all sheet metal parts in the drawing.)

3 distance constraints, 1 coincident constraints, 0 tangent constraints and 0 rigid sets for holes were created.

The constraints can be edited in the Mechanical Browser panel.

Command Options

Option	Description
Settings	Prompts you: **Select setting [Parametrize holes/Back] <Back>** - choose an option. • **Parametrize holes**: Toggles creation rigid sets constraints for internal holes on flanges. Prompts you: Parametrize holes [ON/OFF/Back] <ON> - choose an option or press Enter to accept the current value. • **Back**: Returns to the previous prompt.

Related Commands

MechanicalBrowserOpen - opens the Mechanical Browser panel.
SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmExtrude - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.
SmFlangeBase - creates a base (initial) flange of a sheet metal part from a closed 2D entity.
SmFlangeConnect - closes gaps between two arbitrarily oriented flanges.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlip - Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange.
SmReliefCreate - makes proper corner and bend reliefs.

SmReliefCreate

Makes proper corner and bend reliefs.
Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge.
The command automatically decides whether a relief has to be created for given bend. The size is either automatically chosen by default from the Sheet Metal Settings in the drawing or can be entered by user. In order to force the creation of a bend relief for the given bend, the user has to select the face of desired bend. In this case a bend relief will be created, even if the command decides a bend relief is not needed (i.e. it detects there are no clashes or stretches of the material).
Since V18 command supports creation of reliefs on wrong bends.

Accessing the Command

command bar: *smreliefcreate*
menu bar: *Sheet Metal | Create Relief*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smreliefcreate***
Prompts you in the command bar:

Select a hard edge or bend face, flange face, 3D solid / <Entire model>: (Select an item.)

Select another hard edge or bend face for the corner relief / <Run with the current selection>: (Select another item or right click to execute the command on the current selection.)
Enter relief size by bend radius ratio or [Settings/Auto] <Auto>: (Type a value or choose an option.)

Command Options

Option	Description
Entire model	Creates reliefs on all hard edges, bends and corners in the entire model.
Settings	Allows to access the command settings. Prompts you in the command bar: **Force bend reliefs [ON/OFF/Back] <Back>:** • ON: Forces bend relief creation on bends. Bend reliefs will be created on any bend that does not have orthogonal side faces. • OFF: Bend reliefs on bends will be created only when a tear edge is found. • Back: Returns to the default command prompt
Auto	The command automatically detects bend radii and adjusts the relief size accordingly.

Procedures

Creating missing reliefs

Related Commands

SmBendCreate - converts hard edges (sharp edges between flange faces) and junctions into bends.
SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected faces or entities.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmFlangeSplit - splits a flange along a line drawn on its face.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.

SmReliefSwitch

Converts corner reliefs to circular, rectangular or V-type reliefs. Converts bend reliefs to smooth, round, rip or rectangular bend reliefs. Allows to change the parameters of existing corner reliefs.

Accessing the Command

command bar: *smreliefswitch*
menu bar: *Sheet Metal | Relief | Switch*
toolbar: *Sheet Metal | Relief |*
ribbon: *Sheet Metal | Modify |*

: ***smreliefswitch***
Prompts you in the command bar:

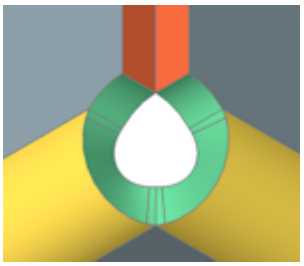
Select face(s), 3d solid(s) to switch relief(s) to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>: (Select a relief face, bend face, flange face or a 3D Solid or press Enter to apply the V-type relief to the entire model; or choose an option.)
Entities in set: 1
Select face(s), 3d solid(s) to switch relief(s) to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>: (Select a relief face, bend face, flange face or a 3D Solid or choose an option or press Enter.)

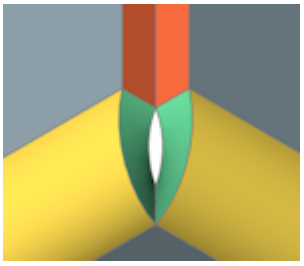
Command Options

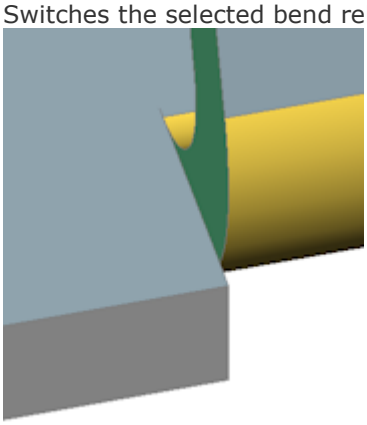
Option	Description

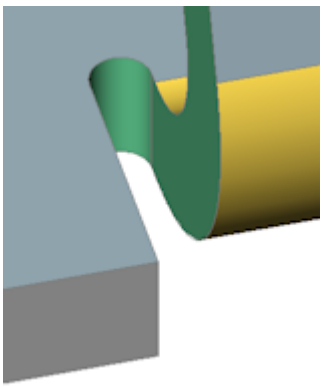
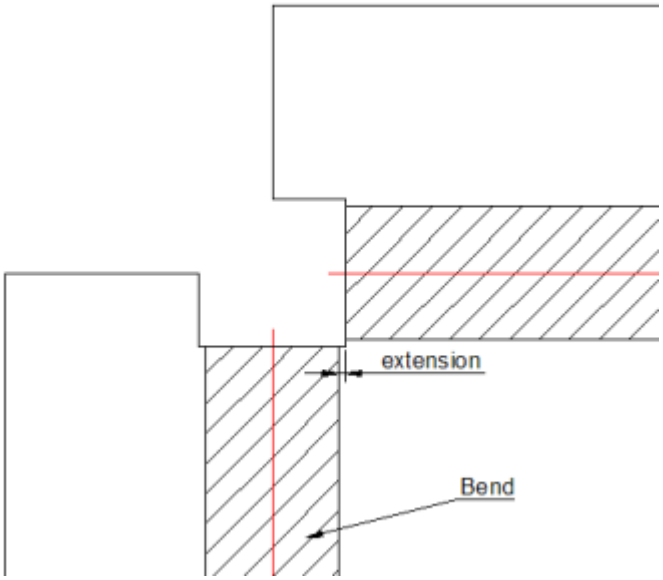
Rectangular (Corner)	Switches the selected corner reliefs to rectangular.
	
	Prompts you:
	Select face(s), 3d solid(s) to switch relief(s) to rectangular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>:
	Do the following:
	• (option) Choose Settings: You are prompted:
	Enter the extension or auto <0.00>:
	Do one of the following:
	  • Type a value
	  • Select Auto to keep the current extension.
	• Select the reliefs to be switched or press Enter to select the entire model.

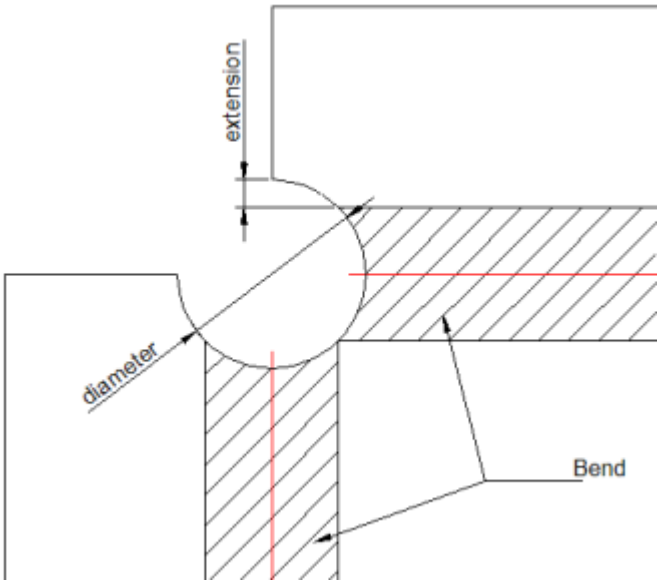
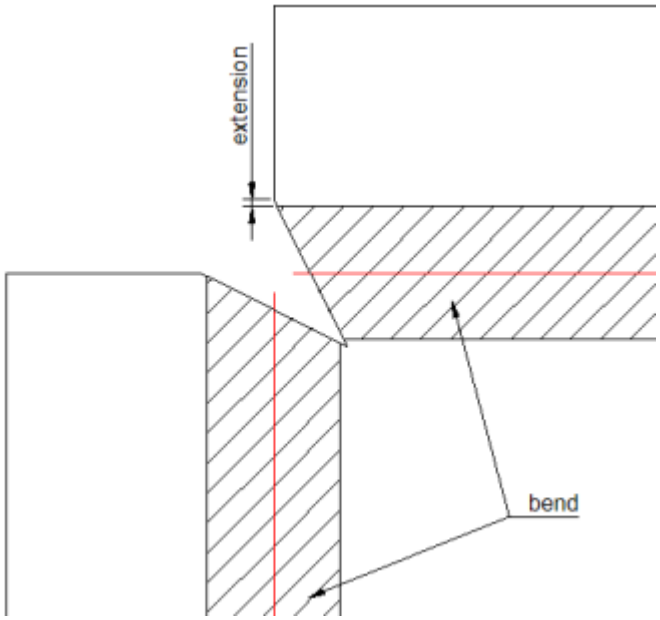
Rectangular (Bend)	Switches the selected bend reliefs to rectangular:

Circular	Switches the selected corner reliefs to circular. Only corner reliefs which are related to exactly two bends can be switched to circular reliefs.  Prompts you: Select face(s), 3d solid(s) to switch relief(s) to circular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>: Do the following: • (option) Choose Settings: You are prompted: Set Extension/Diameter/Reference <Extension>: • Extension:sets the relief extension. Prompts you: Enter the extension or auto <0.00>: Type a value or select Auto to keep the current extension of the selected reliefs. • Diameter:sets the relief diameter. Prompts you: Enter the desired diameter <Auto>: Type a value or select Auto to use the minimal diameter. • Reference:sets the reference for the relief. Prompts you: Select the reference of the relief [Middle/Corner] <Middle>: • Middle: the intersection of the bend lines. • Corner: the circular arc through the base points of the corresponding rectangular reliefs. • Select the reliefs to be switched or press Enter to select the entire model.
V-type	Switches the selected corner reliefs to V-type. Only corner reliefs which are related to exactly two bends can be switched to v-type.

	
	Select face(s), 3d solid(s) to switch relief(s) to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>:
	Do the following:
	• (option) Choose Settings: You are prompted:
	Enter the extension or auto <0.00>:
	Do one of the following:
	  • Type a value
	  • Select Auto to keep the current extension.
	• Select the reliefs to be switched or press Enter to select the entire model.

Smooth	Switches the selected bend reliefs to smooth. Prompts you: Select face(s), 3d solid(s) to switch relief(s) to smooth [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>: Select the bend reliefs to be switched or press Enter to select the entire model. *Smooth* bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration.
Rip	Switches the selected bend reliefs to rip.  Select face(s), 3d solid(s) to switch relief(s) to rip [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>: Select the bend reliefs to be switched or press Enter to select the entire model.

	Rip bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration..
Round	Switches the selected bend reliefs to round.  Select face(s), 3d solid(s) to switch relief(s) to round [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>: Select the bend reliefs to be switched or press Enter to select the entire model. *Round* bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration.
Settings	Allows you to change the parameters which are used to create the relief. The prompt depends on the target type of relief. The following settings are available: rectangular and v-type reliefs: extension  • circular reliefs: minimal extension, diameter of the relief in the unfolded representation, reference to determine the center of the relief in the unfolded representation

	 • V-type reliefs: 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. If this option is chosen as the response to the initial prompt, all corner reliefs switch to V-type.

Procedures

Switching the bend relief type
Switching the corner relief type

Related Commands

SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmFlangeEdge - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.
SmJunctionSwitch - allows changing a symmetrical junction feature to one with overlapping faces.
SmReliefCreate - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge.

SmRepair

Restores the 3D solid model of a sheet metal part by thickening one of its sides: all thickness faces become perpendicular to flange faces.
Fixes problems specific to parts with lofted bend features: merges adjacent bends and provides tangential connection with flanges.
The command is able to convert wrong bends to regular bends. If a single wrong bend, it will be repaired. If a 3D solid is selected, then all the wrong bends will be converted to regular bends.

Accessing the Command

command bar: *smrepair*
menu bar: *Sheet Metal | Repair Sheet Metal*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Heal |*

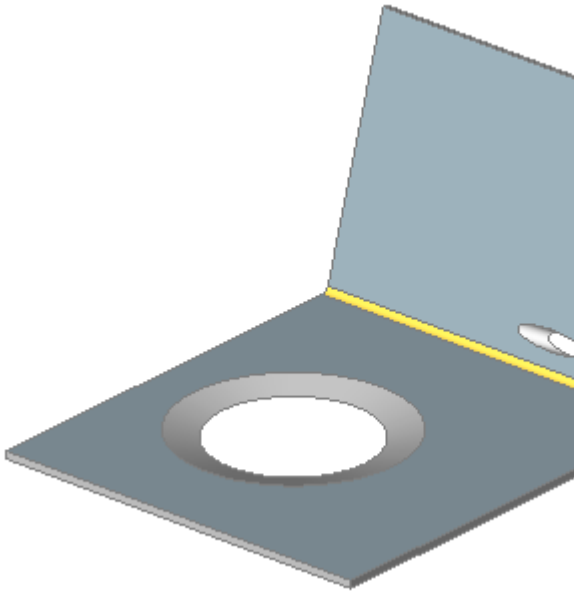
: ***smrepair***
Prompts you in the command bar:

Select a face to repair: (Select a face.)

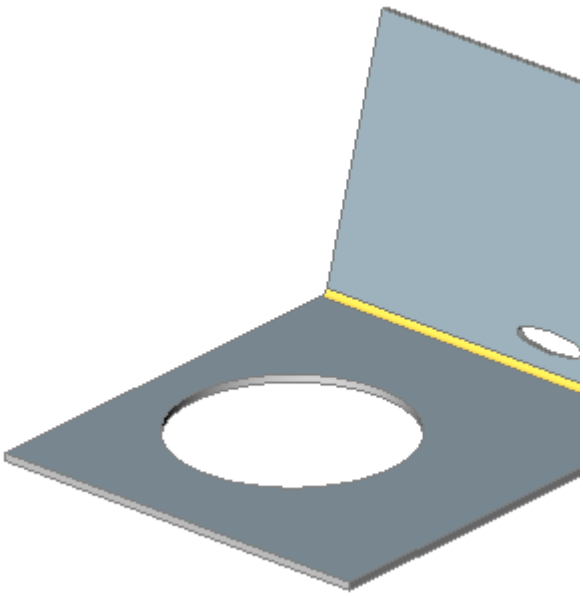
Scan for lost details (may take time)? [Yes/No/Settings] <No>: (Press Enter or choose an option.)

A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created.

Notice that the size and position of the opening on the selected face is preserved; the size and position of the opening in the opposite face is adjusted.

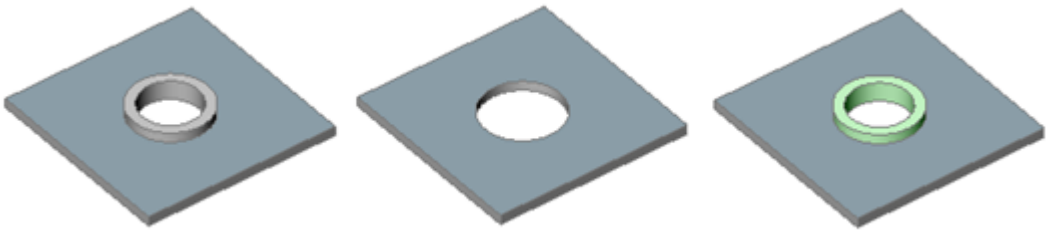
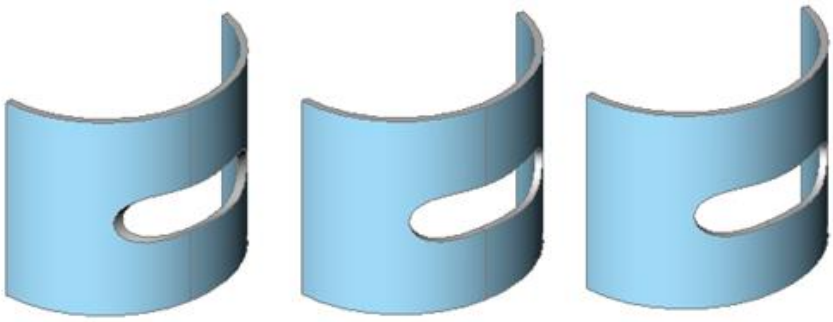


before



after

Command Options

Option	Description
Scan for lost details	Searches and retains as separated bodies model parts which remained after repairing. Prompts you: **Enter scan tolerance <0.01>**: Type a value or press enter to accept the default value. Scan tolerance: The fraction of the initial model volume, which will be ignored, only larger bodies will be retained. E.g. if the scan tolerance = 0.25, only parts with a volume that is larger than 25% of the initial body will be retained.  From left to right: • Initial model with unrecognized form feature • Model after SmRepair without scanning for lost details: the circular feature is removed. • Model after SmRepair with scanning. Two bodies are created: a horizontal flange with a hole and a separate body (highlighted in green color). Bodies can be unified using the Union command if required.
Settings	Prompts you: **Enable lofted bend repair? [Yes/No] <Yes>:** Switch on/off possibility to join adjacent lofted bends  From left to right: • Initial model with two lofted bend features • Model after SmRepair without lofted bend repair: the hole became perpendicular to the features faces. • Model after SmRepair with lofted bend repair: two lofted bend features are united to a single one.

Procedures

To repair a sheet metal body
To repair a sheet metal body with lofted bends

Related Commands

SmConvert - automatically recognizes flanges, bends and lofted bends in a 3D solid.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
SmDissolve - removes sheet metal data from the selected faces or entities.

SmReplace

Replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.

Accessing the Command

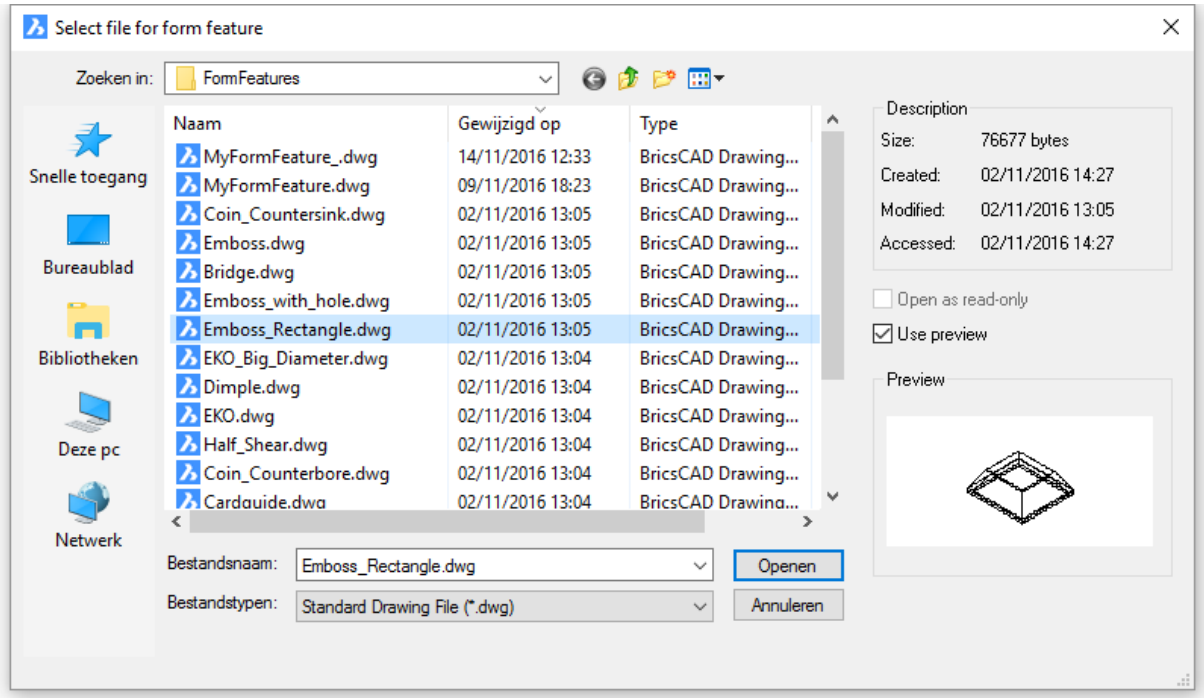
command bar: *smreplace*
menu bar: *SheetMetal | Replace Form Features*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Modify |*

: ***smreplace***
Prompts you in the command bar:
Select several entities/subentities: (Select a form feature.)

Entities in set: 1

Select several entities/subentities: (Select more form features or press Enter to stop selecting.)

A dialog box displays:



(Select a form feature, then press the *Open* button.)

Prompts you:

Replace all similar form features [Yes/No/parametersMode] <Yes>: (Press Enter to replace all similar form features in the model or select an option.)

Command Options

Option	Description
Yes	Replaces all form features of the same type as selected feature(s).
No	Repaces the selected form feature(s) only.
Parameters Mode	Allows to control whether parameter values are inherited from the form features being replaced or not. Prompts you: Select mode of replacement [component Defaults/Inherit from replaced feature/Edit parameters] <Inherit from replaced feature>: • Component Defaults: Parameter values are copied anyway. • Inherit from Replaced Feature: If possible parameter values are copied from the form feature(s) being replaced. • Edit parameters: Allows to edit all parameters. Prompts you: Enter parameter name or press ENTER to finish - Type a parameter name. Enter Expression - Enter an expression.

Related Commands

BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.

SmRibCreate

Creates associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when its defining profile is changed.
The difference with linear rib features inserted from the library by BmInsert is that rib features created by SmRibCreate can have an arbitrary trajectory.

Accessing the Command

command bar: *smribcreate*
toolbar: *Sheet Metal* |
ribbon: *Sheet Metal* | *Modify* |

: ***smribcreate***
Prompts you in the command bar:

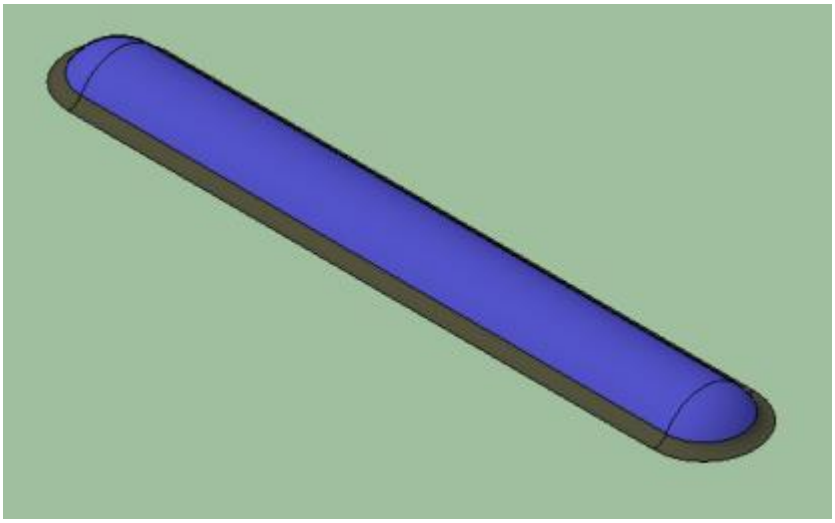
Select flange face: (Select a flange face.)

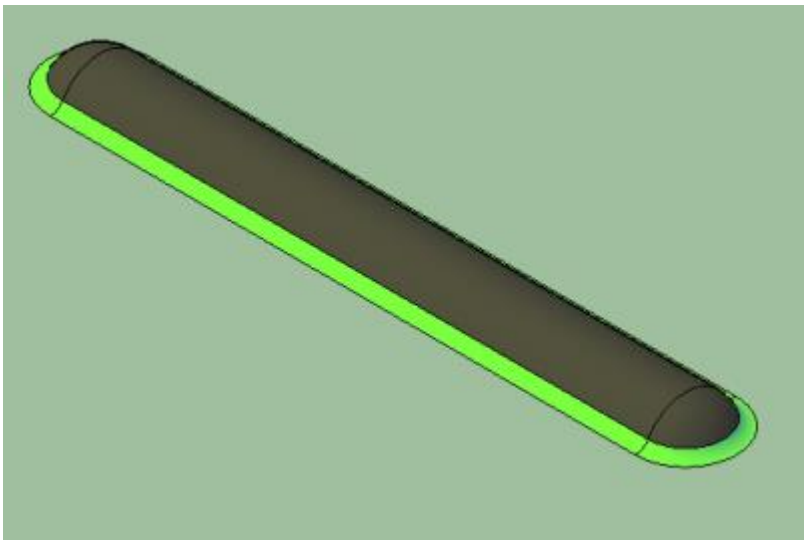
Select curve or [Profile radius/Round radius]: (Select a curve entity or choose an option.)

Entities in set: 1

Select curve or [Profile radius/Round radius]: (Select more curve entities or press Enter.)

Command Options

Option	Description
Select a curve	Select one or more 2D linear entities. lines, polylines, circles, arcs, ellipses, elliptical arcs and splines are accepted. The 2D curves can be open or closed but cannot intersect itself or other form features.
Profile Radius	Sets the radius of the rib profile (blue face in the image below). The initial value of the Profile Radius is taken from the SMDEFAULTRIBPROFILERADIUS system variable. 

Round Radius	Sets the radius of the smoothing fillets (green faces in the image below). The initial value of the Round Radius is taken from the SMDEFAULTRIBROUNDRADIUS system variable. 

Related Commands

BmInsert - inserts an existing mechanical component or sheet metal form feature into the current drawing.
SmConvert - automatically recognizes flanges and bends in a 3D solid.

SmSelect

Selects hard edges or form features.

Accessing the Command

command bar: *smselect*
menu bar: *Sheet Metal* | *Select*
toolbar: *Sheet Metal* | or
ribbon: *Sheet Metal* | *Select* | or

: ***smselect***
Prompts you:
Select the option [Hard edges/Same form features/SImilar form features]: (Choose an option.)

Command Options

Option	Description
Hard edges	Selects all hard edges in the model.
Same form features	Selects all form features which are identical to the selected feature(s). Prompts you: Select form feature faces - select a form feature face. Entities in set: 1 Select form feature faces - select more form feature faces or press Enter.
Similar form features	Selects all form features which are similar (= same type, but different parameter values) to the selected feature(s) . Prompts you: Select form feature faces - select a form feature face. Entities in set: 1 Select form feature faces - select more form feature faces or press Enter.

Related Commands

dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
dmSelectEdges - selects the edges of faces and solids.
SmBendCreate - converts hard edges (sharp edges between flange faces) and junctions into bends.
SmDelete - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.dmSelect - selects edges and faces of 3D solids or surfaces based on their geometric properties.
SmJunctionCreate - converts hard edges (sharp edges between flange faces) and bends into junctions.

SmUnfold

Creates an unfolded 2D or 3Drepresentation of a sheet metal part.

Accessing the Command

command bar: *smunfold*
menu bar: *Sheet Metal | Unfold Body*
toolbar: *Sheet Metal |*
ribbon: *Sheet Metal | Flatten |*

: ***smunfold***
Prompts you in the command bar:
Select a flange or lofted bend face to start unfolding [Settings]: (Select the base flange of the sheet metal part.)

Select position of the unfolded body: (Specify a point in the drawing.)

Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>: (Press Enter to keep the unfolded body in the drawing or choose an option.)

Command Options

Option	Description

Settings	Allows to override the default unfolding settings.
	Prompts you:
	Select [lofted bend Tolerance/Form feature behavior] <Form feature behaviour> - select an option.
	• Lofted bend tolerance: Allows to specify the tolerance for lofted bends. Prompts you: Set tolerance for lofted bends straight edges determination - type a value.
	• Form feature behavior: Allows to modify the value of the SMFORMFEATUREUNFOLDMODE system variable. Prompts you: Select form feature behaviour [Keep/Remove/Project/Contour/Symbol] <Symbol> - select an option.
	  • Keep: Keeps the form features geometry on the unfolded representation.
	  • Remove: Removes the form features geometry from the unfolded representation.
	  • Project: Generates a form feature symbol by projecting the form feature edges on the flange plane.
	  • Contour: Generates a form feature symbol by projecting an external contour and holes of the form feature on the flange plane.
	  • Symbol: Copies the unfold symbol on layer UNFOLD_SYMBOL_UP or layer UNFOLD_SYMBOL_DOWN depending on the form feature direction) to the unfolded representation.

Keep	Places the unfolded 3D solid in the current drawing.

Save 2D geometry	Saves the 2D geometry of the unfolded metal part as a separate drawing file. Displays a Save File dialog window. • Select a folder. • Type a name in the *File Name* field. • Select either *Drawing Exchange Format (*.dxf)* or *Standard Drawing File (*.dwg)* from the *Save as type* list button. • Click the *Save* button to create the file.
Save 3D geometry	Saves the 3D solid of the unfolded metal part as a separate drawing file. Displays a Save File dialog window. • Select a folder. • Type a name in the *File Name* field. • Select either *Drawing Exchange Format (*.dxf)* or *Standard Drawing File (*.dwg)* from the *Save as type* list button. • Click the *Save* button to create the file.
Optimize Bend Annotations	Optimizes the bend annotations in the exported geometry.

Procedures

To unfold the sheet metal body

Related Commands

SmAssemblyExport - traverses the Assembly Structure of the current drawing, searches for 3D solids in its component, and applies the SmConvert and SmExport2d commands.
SmExport2d - exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
SmExportOSM - exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system).

T

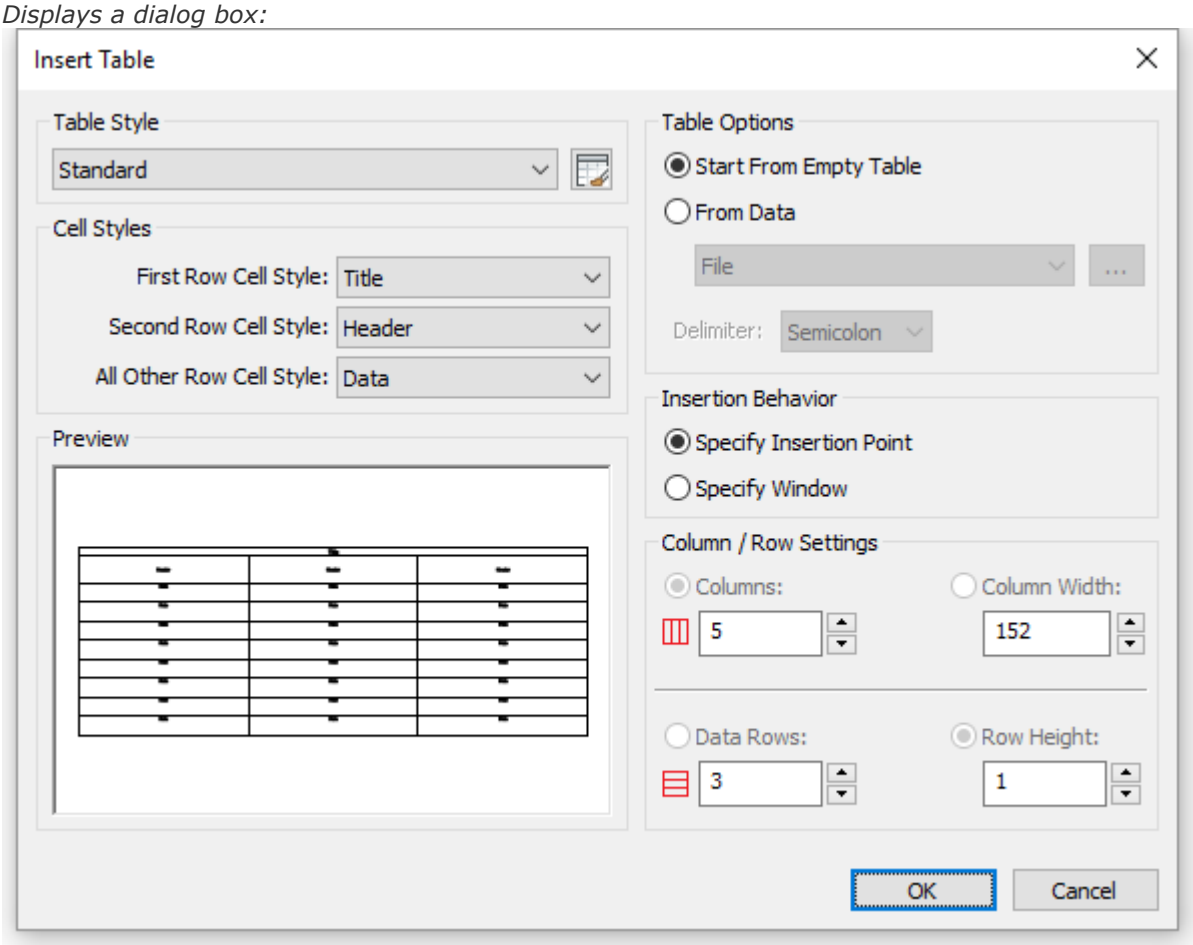
Table

Creates a table entity in the drawing through a dialog box.

Accessing the Command

command bar: *table*
menu bar: *Draw | Table...*
toolbar: *Draw |*

: ***table***
Displays a dialog box:



Choose options, and then click OK.

Command Options

Option	Description
Table Style	Click the button, then select a table style.
Table Styles Explorer	Opens the Drawing Explorer - Table Styles dialog (see the TableStyle command)
Cell Styles	Choose the cell styles. The options are: *Title*, *Header* or *Data*. The properties of the cell styles are defined in the table style.
First Row Cell Style	Click the button, then select a cell style for the first row. Default: *Title*

Second Row Cell Style	Click the button, then select a cell style for the second row. Default: *Header*
All Other Row Cell Styles	Click the button, then select a cell style for the other rows. Default: *Data*
Table Options	
Start From Empty Table	Creates a new empty table.
From Data	The cell content is read from a data file. 1. Click the *Browse* button (...). A *File* dialog displays. 2. Do one of the following: • Choose *Delimiter Separated format (*.csv)* from the *File of type* option button, then select a *CSV* file. • Choose *XML Spreadsheet format (*.xml)* from the *File of type* option button, then select an *Excel XML Spreadsheet 2003 (*.xml)* file.
Delimiter	Allows to specify the delimiter character to separate the data in the source file. The options are: *Semicolon*, *Comma*, *Space* and *Tab*.
Insertion Behavior	
Specify Insertion Point	When the OK button is pressed you are prompted to specify an insertion point for the table. The top left corner of the table is attached to the cross hairs.
Specify Window	When the OK button is pressed, prompts you in the command bar: Specify first corner: Enter the position of the top left corner of the table. Specify second corner: Enter the position of the bottom right corner of the table.
Columns / Rows Settings	Specifies the number of columns and data rows and/or the column width and row height.
Columns	Enter the number of columns.
Column Width	Specify the column width.
Data Rows	Enter the number of data rows.
Row Height	Specify the row height.
OK	Inserts the table in the drawing (See Insertion Behavior).

Procedures

To create an empty table
To create a table from data
To create a table from an Excel spreadsheet
General procedure to create fields

Related Commands

-Table - creates a table entity in the drawing through the command bar.
Tablemod - modifies the properties of a selection of table cells.
Tabledit - edits the cell content.
TableExport - exports the content of a table entity in CSV file format.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.
Field - inserts a field in a multiline text, table cell or attribute.

-Table

Creates a table entity in the drawing through the command bar.

Accessing the Command

command bar: *-table*

: ***-table***
Prompts you in the command bar:

Enter number of columns or [Auto] <4>: specify the number of columns or type A and press Enter.

Enter number of rows or [Auto] <4>: specify the number of data rows or type A and press Enter.

Specify insertion point or [Style/Width/Height]: specify the insertion point or choose an option.

Specify first corner: Enter the position of the top left corner of the table.

Specify second corner: Enter the position of the bottom right corner of the table.

Command Options

Option	Description
Auto	The number of columns or rows is defined dynamically when inserting the table. Prompts you in the command bar: Specify first corner: Enter the position of the top left corner of the table. Specify second corner: Enter the position of the bottom right corner of the table.
Style	Specifies the Table Style. Prompts you in the command bar: Enter table style name <Standard>: type the name of the table style, then press Enter.
Width	Specifies the column width. Prompts you in the command bar: Enter cell width <6.0000>: type the column width in the command bar, then press Enter.
Height	Specifies the minimum row height. Prompts you in the command bar: Enter minimum cell height <4>: type the minimum row height in the command bar, then press Enter.
Specify insertion point	Specify the position of the top left corner of the table.

Procedures

To create an empty table
To create a table from data
To create a table from an Excel spreadsheet
General procedure to create fields

Related Commands

Table - creates a table entity in the drawing through a dialog box.
Tablemod - modifies the properties of a selection of table cells.
Tabledit - edits the cell content.
TableExport - exports the content of a table entity in CSV file format.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

Tabledit

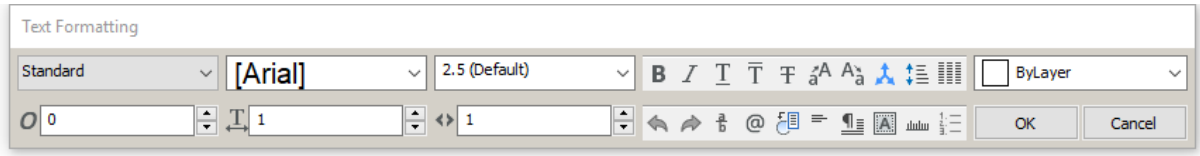
Edits texts in a table cell.

Accessing the Command

command bar: *tabledit*
mouse: right click a table cell (long right click if the Quad is active) or double click a table cell to immediately open the Multiline Text dialog box.

: ***tabledit***
Prompts you in the command bar:
Pick table cell to edit: click a table cell.

Displays the Multiline Text dialog box:



Command Options

See Mtext.

Procedures

To edit the table content
Editing a table in the Properties bar
Editing a cell in the Properties bar

Related Commands

Table - creates a table entity in the drawing through a dialog box.
-Table - creates a table entity in the drawing through the command bar.
Tablemod - modifies the properties of a selection of table cells.
TableExport - exports the content of a table entity in CSV file format.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.
Mtext - places multi-line text in drawings through a dialog box.
Ddedit - edits single-line text, multi-line text, attribute definitions, and attribute text.

TableExport

Exports the content of a table entity in CSV file format.

Accessing the Command

command bar: *tableexport*
context menu: *Export Table...*

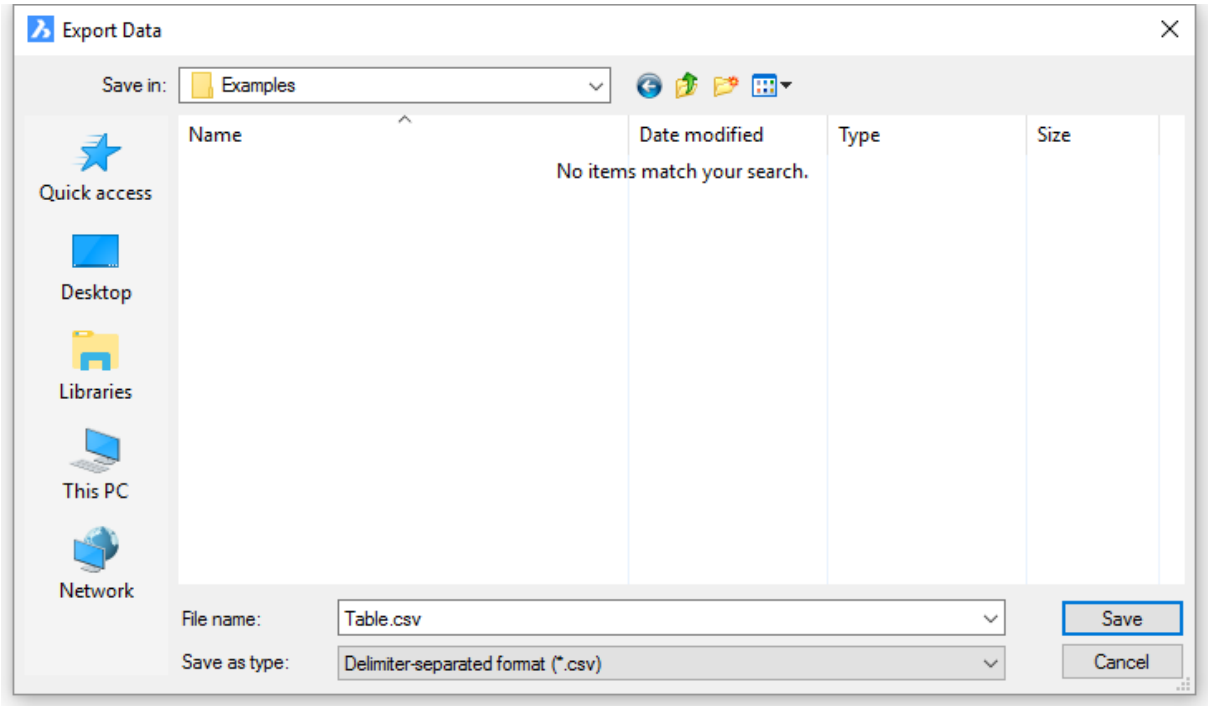
: ***tableexport***

Prompts you in the command bar:

Select Table to export: click a table entity

The Export Data dialog box displays:

If started from the context menu, the Export Data dialog displays right away.



Command Options

Option	Description
Save in:	Select a folder.
File name:	Type a name for the CSV file. Default name: <table_name>.csv
Save as type	Allows to choose the format of the export file. The options are: • Delimiter-separated format (*.csv) • XML Spreadsheet format (*.xml)
Save	Exports the cell data and creates the export file.
Cancel	Aborts the TableExport command.

Procedures

To export a table
Defining the list separator character

Related Commands

Table - creates a table entity in the drawing through a dialog box.
-Table - creates a table entity in the drawing through the command bar.
Tablemod - modifies the properties of a selection of table cells.
Tabledit - edits the cell content.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

TableMod

Modifies the properties of a selection of table cells.

Accessing the Command

command bar: *tablemod*
mouse: click a table cell

: ***tablemod***

Prompts you in the command bar:
Pick table cell to edit: click the cell you want to modify or press and hold the left mouse button then define a rectangle to select multiple cells.

Displays a toolbar:

Command Options

Option	Description
insert row above	Inserts an empty row above the selection.
insert row below	Inserts an empty row below the selection.
delete row	Deletes the row(s) of the selection.
insert column left	Inserts an empty column at the left of the selection.
insert column right	Inserts an empty column at the right of the selection.
delete column	Deletes the column(s) of the selection.
merge cells	Merges the selected cells. Choose an option: Merge All Merge by Row Merge by Column
unmerge cells	Splits previously merged cells.
border style	Sets the border style of the selection (not implemented yet).
text alignment	Sets the text alignment of the selection. Choose an option:

	Top Left Top Center Top Right Middle Left Middle Center Middle Right Bottom Left Bottom Center Bottom Right
lock / unlock	Locks / Unlocks the selection. Click the tool button. BricsCAD responds in the command bar: **Cell contents locked** **Cell contents unlocked**
cell format	Sets the cell format of the selection (not implemented yet)
Title cell style	Sets the style of the selected cells. Choose an option: Title Header Data
ByEntity cell background	Sets the background color of the selection. Choose a color, or click *Select Color...*, to open the Index Color palet.
Right click	Displays a context menu: Delete Cell Content Clear Cell Overrides Clear Table Overrides • Delete Cell Content: deletes the text in the selected cells. • Clear Cell Overrides: resets all cell properties of the selected cells to the Cell Style settings as defined in the Table Style. • Clear Table Overrides: resets all cell properties of all cells to the Cell Style settings as defined in the Table Style.

Procedures

Editing a table using grips
Editing a table using the Table toolbar
Editing a table in the Properties bar
Editing a cell in the Properties bar

Related Commands

Table - creates a table entity in the drawing through a dialog box.
-Table - creates a table entity in the drawing through the command bar.
Tabledit - edits the cell content.
TableExport - exports the content of a table entity in CSV file format.
TableStyle - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

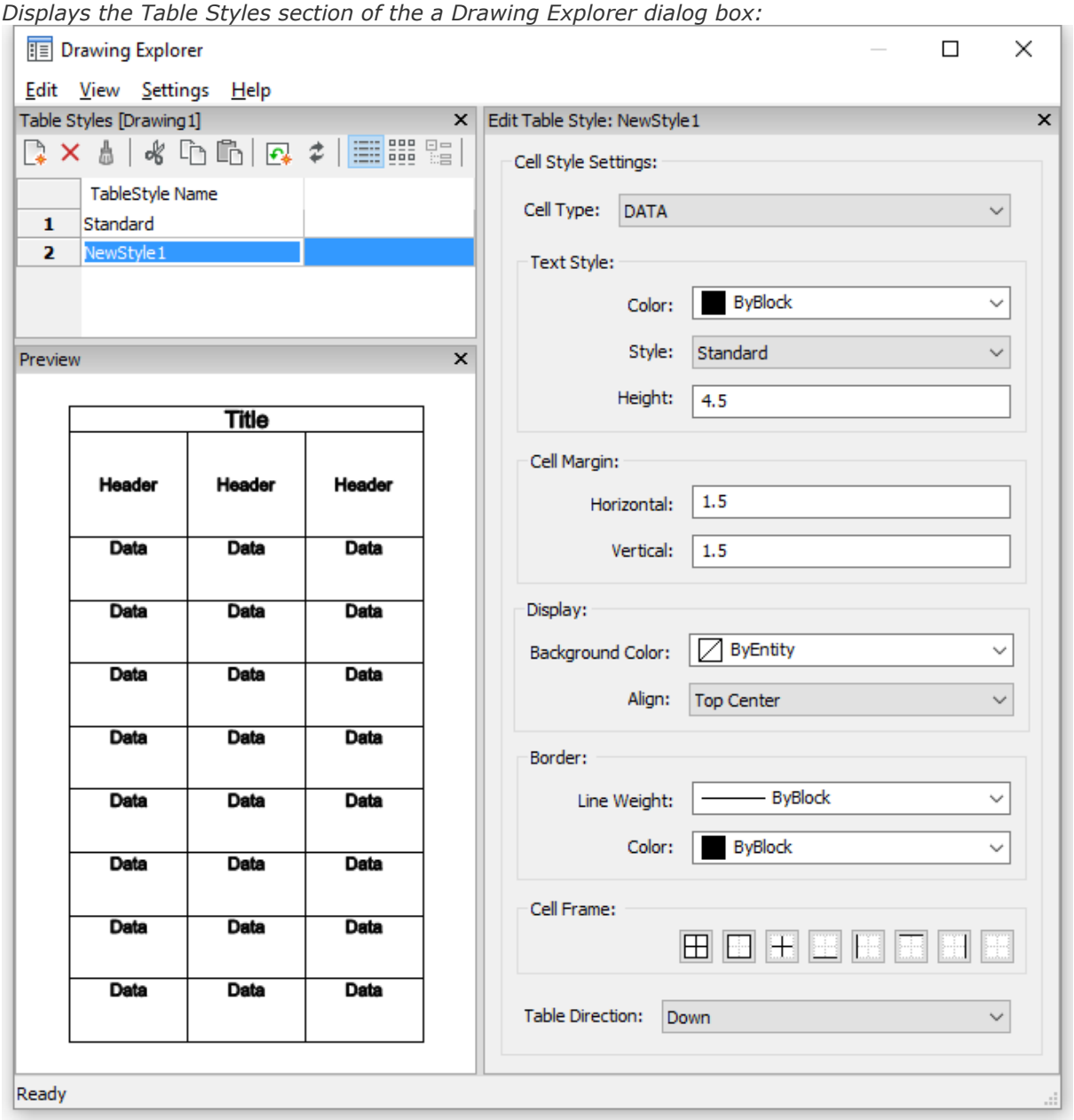
TableStyle

Creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

Accessing the Command

command bar: *tablestyle*
menu bar: *Settings | Table Styles...*

: ***tablestyle***
Displays the Table Styles section of the a Drawing Explorer dialog box:



Command Options

Option	Description
New	Creates a new table style. TableStyle Name 1 NewStyle1 Click the *TableStyle Name* field to replace the *NewStyle1* default name.
Delete	Deletes the currently selected table style. A table style that is being used in the drawing cannot be deleted.
Preview	A preview of the currently selected table style.
Cell Style Settings	Defines the cell style settings for each cell style (*Title*, *Header*, *Data*)
Cell Type	Press the button, then choose a cell style. DATA HEADER TITLE The settings for the selected cell style display.
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: distance between the cell content and the vertical cell borders. • Vertical: distance between the cell content and the horizontal cell borders
Display	Defines the background color and the alignment. • Background Color: pick a color or choose *Select Color...* to display the *Color* palette. • Align: press the list button, then pick an alignment: Top Left Top Center Top Right Middle Left Middle Center Middle Right Bottom Left Bottom Center Bottom Right
Border	Sets the lineweight and color of the cell borders. • Line Weight: pick a line weight in the list. Please notice line weights only display if the *LWT* option in the Status Bar is active. • Color: pick a color or choose *Select Color...* to display the *Color* palette.
Cell Frame	Click a button to choose an option:
Table Direction	Defines how the table is organized. The options are: • Down: Title and header cells on top. • Up: Title and header cells at the bottom.

Procedures

To open the Table Styles Explorer
To create a table style
To edit a table style
To apply a style to a table
To delete a table style

Related Commands

-Table - creates a table entity in the drawing through the command bar.
Table - creates a table entity in the drawing through a dialog box.
Tablemod - modifies the properties of a selection of table cells.
Tabledit - edits the cell content.
TableExport - exports the content of a table entity in CSV file format.
Color - specifies the current color, through a dialog box.
Style - creates and modifies text styles with the Drawing Explorer.
-Style - creates and modifies text styles in the command bar.

Explorer - Opens the Drawing Explorer dialog box.

Tablet

Configures and calibrates tablets, and toggles tablet mode.

Accessing the Command

command bar: *tablet*
shortcut: *T*
alias: *ta*
menu bar: *Settings | Tablet*
status bar: *TABLET*

: ***tablet***
Prompts you in the command bar:

Tablet: [ON/OFF/CALibrate/ConFiGure]<On>: (Enter an option.)

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer.

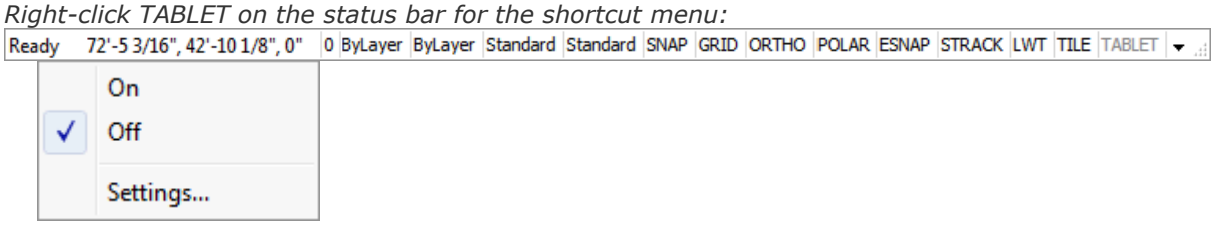
Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows\System32.

Command Options

Option	Description
ON	Turns on tablet digitizing mode. The entire surface of the table is used like a mouse for picking points and selecting entities.
OFF	Turns off tablet digitizing mode. The menu areas become available for selecting commands.
CALibrate	Calibrates the coordinates of the tablet.
ConFiGure	Configures the tablet's menu areas.

Status Bar Menu

Right-click TABLET on the status bar for the shortcut menu:



Shortcut Menu	Description
ON	Turns on tablet digitizing mode.
OFF	Turns off tablet digitizing mode.
Settings	Opens the Coordinate Input section of the Settings dialog box. ⊟ **Drawing** ⊟ **Drafting** ⊞ **Drawing units** ⊟ **Coordinate input** Orthogonal mode — ☐ Orthogonal mode Tablet mode — ☐ Use tablet
Tablet mode	Toggles tablet mode: • On - switches tablet to digitizing mode. • Off - switches tablet to command selection mode.

Procedures

To use a digitizing tablet in BricsCAD
To download the tablet overlay and CUI files
To load the overlay CUI file
To switch the tablet ON / OFF
To configure the tablet
To calibrate the tablet

Related Commands

None.

TabSurf

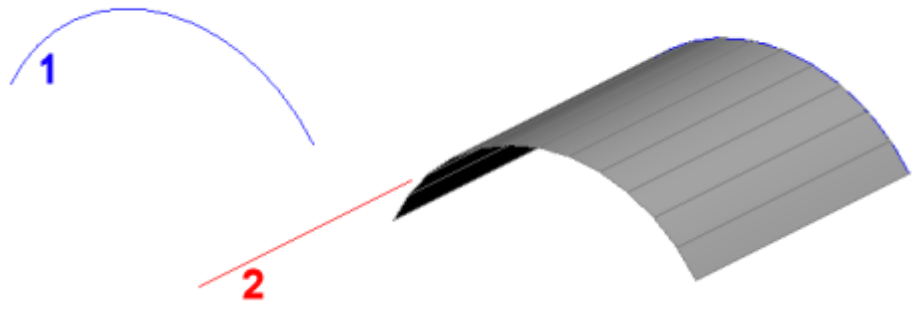
Creates a 3D mesh surface by projecting an entity along a path.

Accessing the Command

command bar: *tabsurf*
menu bar: *Model | Meshes | Extruded Surface*
toolbar: *Meshes | 3D Surfaces |*

: ***tabsurf***

Prompts you in the command bar:
Select entity to extrude: (Choose a single open entity.) (1)
Select line or open polyline for extrusion path: (Pick a line or polyline.) (2)



The entity to be extruded must be open and must be a single entity.

The extrusion path can be a line or open polyline. The polyline need not be a line segment; if a polyarc or multisegment polyline, the extrusion, however, is always "straight," going from the start to the endpoint of the polyline.

The path is a vector, which means its length specifies the length of the surface; its direction specifies the direction of extrusion. The extrusion starts at the entity to be extruded, but goes in the direction indicated by the path.

The end selected on the polyline or line defines the direction of the extrusion.

Arc segments of the extruded entity are divided in a number of equal length intervals defined by the SURFTAB1 system variable.

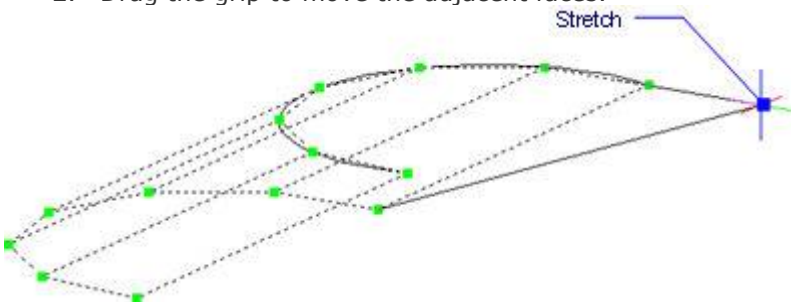
Command Options

Option	Description
Select entity to extrude	Specifies the cross-section shape of the extrusion.
Select line or open polyline for extrusion path	Specifies the length and direction of the extrusion.

Grips Editing

Mesh-formed tabulated surfaces can be edited directly through grips:

1. Select the tabsurf. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.



Related Commands

Ai_Box - draws 3D boxes as mesh surfaces.
Ai_Cone - draws 3D cones as mesh surfaces.
Ai_Cylinder - draws 3D cylinders as mesh surfaces.
Ai_Dish - draws 3D dishes as mesh surfaces.
Ai_Dome - draws 3D domes as mesh surfaces.
EdgeSurf - draws edge surfaces as mesh surfaces.
Ai_Pyramid - draws 3D pyramids as mesh surfaces.
RevSurf - draws revolved surfaces as mesh surfaces.
RuleSurf - draws ruled surfaces as mesh surfaces.
Ai_Sphere - draws 3D spheres as mesh surfaces.
Ai_Torus - draws 3D donuts as mesh surfaces.
Ai_Wedge - draws 3D wedges as mesh surfaces.

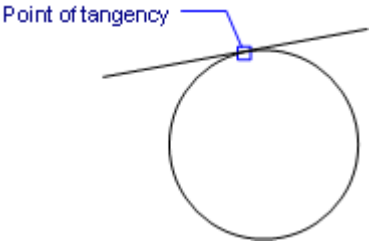
Tangent

Toggles tangent entity snap; snaps to the tangency of circles, arcs, ellipses and elliptical arcs.

Accessing the Command

command bar: *tangent*
transparent: *tan*
toolbar: *Entity Snaps* |

: ***tangent***
Displays no prompts in the command bar.



When tangent entity snap is on, the cursor snaps to points of tangency along curved entities, such as circles, arcs, ellipses and elliptical arcs.

Command Options

Option	Description
Tangent	Entering the command toggles Tangent entity snap. It snaps to points of tangency. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap - toggles entity snap modes in the Settings dialog box.
-OSnap - toggles entity snap modes in the command bar.
Aperture - sizes the aperture cursor for entity snaps.

2dIntersection - toggles apparent intersection snap mode.
3dIntersection - toggles intersection snap mode.
Apparent - toggles apparent intersection snap mode.
Center - toggles center snap mode.
Endpoint - toggles endpoint snap mode.
Extension - toggles extension snap mode.
From - allows to enter relative coordinates with respect to a reference point.
GCenter - toggles Geometric Center (GCenter) entity snap.
Insertion - toggles insertion snap mode.
Intersection - toggles intersection snap mode.
Midpoint - toggles midpoint snap mode.
Mtp - allows to specify a point as the midpoint between two points.
Nearest - toggles nearest snap mode.
Node - toggles node (point) snap mode.
None - turns off all entity snap modes.
Parallel - toggles parallel snap mode.
Perpendicular - toggles perpendicular snap mode.
Quadrant - toggles quadrant snap mode.

TemplateFolder

Opens the Template folder in a File dialog box.

Accessing the Command

command bar: *templatefolder*

: ***templatefolder***
Displays a file dialog with box the Template folder open.

The default Template folder is:
C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD\Vxx\en_US\Templates
The template folder path is saved through the TemplatePath user preference.

Command Options

No Options

Related Commands

SupportFolder - opens the main Support folder in a File dialog box.

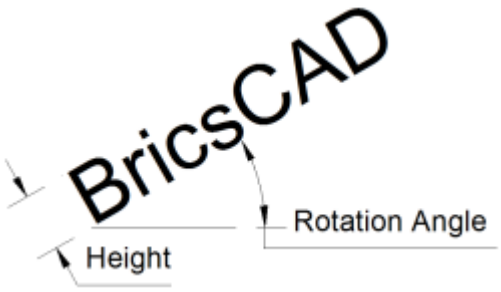
Text

Places single lines of text in the drawing.

Accessing the Command

command bar: *text*
aliases: *tx, dtext*
menu bar: *Draw | Text*
toolbar: *Draw |* A

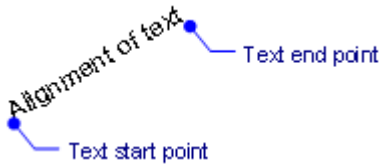
: ***text***
Prompts you in the command bar:
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point, or enter an option.)
Height of text <0.2000>: (Specify a height, or press Enter to accept the default.)
Rotation angle of text <0>: (Specify an angle, or press Enter to accept the default.)
Text: (Enter text, and then press Enter.)
Text: (Enter more text, and then press Enter.)
Text: (Press Enter to exit the command.)



For more control over text properties, use the **MText** command.
To check the spelling of text, use the **Spell** command.
To edit text, use the **DdEdit** command.

Command Options

Option	Description
Start point	Specifies the starting point of the text. Pick a point, or enter x,y coordinates. The starting point is affected by the justification.
Style	Specifies the name of the text style; prompts you: **Text style to use (or '?')** - enter the name of a style or type ?. See Style command.
?	Lists the names of styles found in the drawing; prompts you: **Text style(s) to list** - specifies the styles to list. • * - lists all styles. • name* - lists all names that begin with name. • name - lists the style named name.

Align	Specifies align justification. Aligns text between two points; prompts you: **Text start point** - specify the staring point of the text. **Text end point** - specify the end point for the text. This option supresses the height and angle prompts, because the height and angle are determined by the two pick points. • Rotation angle of text is determined by the angle between the two points. • Height of text is determined proportions. More characters in the same distance make the text shorter. 
Fit	Specifies fit justification. Fits text between two points and prompts for the height; prompts you: **Text start point** - specify the staring point of the text. **Text end point** - specify the end point for the text. **Height of text** - specify the text height. This option supresses 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. Use this justification to force text to fit in an area, such as a closet.
Center	Specifies center justification. Centers text in the middle of the baseline; prompts you: **Center point of text** - specify the center point on the baseline.

Middle	Specifies middle justification. Centers text on the middle of the text; prompts you: **Middle point of text** - specify the middle point.
Right	Specifies right justification. Right-justifies each line of text; prompts you: **Right point of text** - specify the right point on the baseline.
Justify	Displays all justification modes; prompts you: **Text:** **Style/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR/< Start point>** - enter an option. • **TL** - top left • **TC** - top center • **TR** - top right • **ML** - middle left • **MC** - middle center (equivalent to Middle). • **MR** - middle right • **BL** - bottom left (equivalent to Start Point). • **BC** - bottom center (equivalent to Center). • **BR** - bottom right (equivalent to Right).
Height of text	Specifies the height of the text. Enter a number or pick two points. This prompt is not displayed under the following conditions: • Style has set the text to a height other than 0. • Justification mode is Align. If the text style is annotative the height value defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

Rotation angle of text	Specifies the angle of the text. Enter a number or pick two points. This prompt is not displayed when the justification is set to Align or Fit.
Text	Specifies the text. Enter a line of text, and then press Enter. The prompt repeats; enter more text. Each additional line of text is positioned under the previous line. First line of text. Press Enter Next line of text. Press Enter Press Enter to exit command. Exit the command by pressing Enter at the 'Text' prompt.

Procedures

To create single line text
To create multiline text
Defining text system variables

Grips Editing

Text can be moved directly through grips:

1. Select the text face. Notice that it has one grip at its insertion point.
2. Drag the grip to move the text.

Aligned and fitted text have two grips:

- One grip moves the text.
- The second grip stretches and rotates the text. Aligned text also changes its height.

Related Commands

-Text - places text and evaluates LISP expressions.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
Change - changes the position of text.
QText - toggles between text and rectangular representations.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Find - finds and replaces text.
Properties - modifies the properties of text.

-Text

Places single lines of text in the drawing, and optionally evaluates LISP expressions.

Accessing the Command

command bar: *-text*
alias: *-t*

: ***-text***

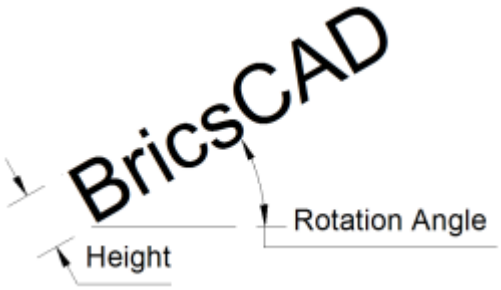
Prompts you in the command bar:

Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point, or enter an option.)

Height of text <0.2000>: (Specify a height, or press Enter to accept the default.)

Rotation angle of text <0>: (Specify an angle, or press Enter to accept the default.)

Text: (Enter text, and then press Enter.)



When the *TextEval* system variable is set to 1, this command evaluates LISP expressions:

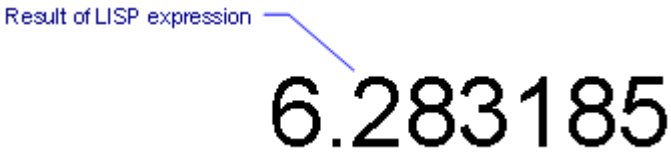
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point.)

Height of text <0.2000>: (Press Enter.)

Rotation angle of text <0>: (Press Enter.)

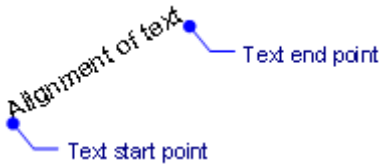
Text: (* pi 2)

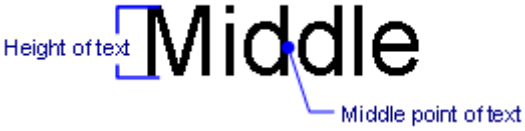
The result of the equation (pi x 2) is placed as text:



Command Options

Option	Description
Start point	Specifies the starting point of the text. Pick a point, or enter x,y coordinates. The starting point is affected by the justification.
Style	Specifies the name of the text style; prompts you: Text style to use (or '?') - enter the name of a style or type ?. See Style command.

?	Lists the names of styles found in the drawing; prompts you: Text style(s) to list - specifies the styles to list. • * - lists all styles. • name* - lists all names that begin with name. • name - lists the style named name.
Align	Specifies align justification. Aligns text between two points; prompts you: Text start point - specify the staring point of the text. Text end point - specify the end point for the text. This option supresses the height and angle prompts, because the height and angle are determined by the two pick points. • Rotation angle of text is determined by the angle between the two points. • Height of text is determined proportions. More characters in the same distance make the text shorter. 
Fit	Specifies fit justification. Fits text between two points and prompts for the height; prompts you: Text start point - specify the staring point of the text. Text end point - specify the end point for the text. Height of text - specify the text height. This option supresses 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. Use this justification to force text to fit in an area, such as a closet.

Center	Specifies center justification. Centers text in the middle of the baseline; prompts you: Center point of text - specify the center point on the baseline.
Middle	Specifies middle justification. Centers text on the middle of the text; prompts you: Middle point of text - specify the middle point. 
Right	Specifies right justification. Right-justifies each line of text; prompts you: Right point of text - specify the right point on the baseline.
Justify	Displays all justification modes; prompts you: Text: Style/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR/<Start point> - enter an option. • TL - top left • TC - top center • TR - top right • ML - middle left • MC - middle center (equivalent to Middle). • MR - middle right • BL - bottom left (equivalent to Start Point). • BC - bottom center (equivalent to Center). • BR - bottom right (equivalent to Right).

Height of text	Specifies the height of the text. Enter a number or pick two points. This prompt is not displayed under the following conditions: • Style has set the text to a height other than 0. • Justification mode is Align. If the text style is annotative the height value defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.
Rotation angle of text	Specifies the angle of the text. Enter a number or pick two points. This prompt is not displayed when the justification is set to Align or Fit.
Text	Specifies the text. Enter a line of text, and then press Enter. The prompt repeats; enter more text. Each additional line of text is positioned under the previous line. First line of text. Press Enter Next line of text. Press Enter Press Enter to exit command. Exit the command by pressing Enter at the 'Text' prompt.

Procedures

To create single line text
To create multiline text
Defining text system variables

Grips Editing

Text can be moved directly through grips:

1. Select the text face. Notice that it has one grip at its insertion point.
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Aligned and fitted text have two grips:

- One grip moves the text.
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Related Commands

Text - places single-lines of text in drawings.
MText and -MText - places paragraph text.
AttDef - defines attribute text.
DdEdit - edits text.
Style - creates and modifies text styles through the Drawing Explorer.
-Style - creates and modifies text styles through the command bar.
QText - toggles between text and rectangular representations.
Change - changes the position of text.
TxtExp - explodes text into polylines.
Spell - checks the spelling of text in drawings.
Properties - modifies the properties of text.
Find - finds and replaces text.

TextScr

Displays the Prompt History window (short for "text screen"); this window displays the last 256 lines of command and prompt text.

Accessing the Command

command bar: *textscr*
transparent: *'textscr*
shortcut: *F2*
menu bar: *View | Prompt History Window*

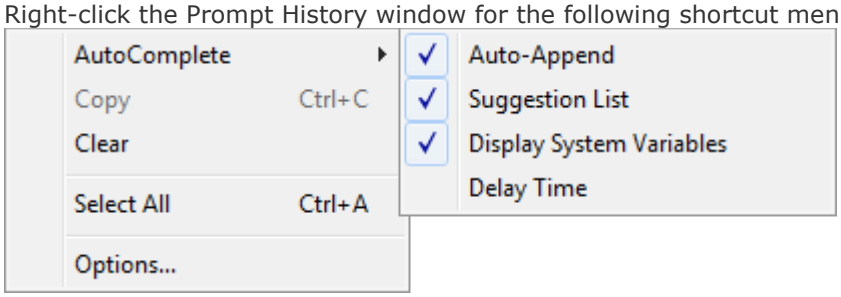
: ***textscr***

Displays window:

To close the window, click the X (in upper right corner) press the F2 function key or use the **GraphScr** command.

Shortcut Menu Options

Right-click the Prompt History window for the following shortcut menu:



Option	Description
AutoComplete	Sets the autocomplete options. See the AutoComplete command.
Copy History	Copies all of the text to the Clipboard.
Copy Last Line	Copies the last line of text to the Clipboard.
Paste	Pastes text in the Command Bar. This option is available only when the Clipboard contains text.
Options	Displays the Display Section of the Options dialog box: Scroll History - specifies the number of lines of text to remember.

Related Commands

AutoComplete - sets the command line autocomplete mode.
CommandLine - displays the command bar.
CommandLineHide - hides the command bar.
GraphScr - brings the graphics window to the foreground.

TextToFront

Sets the draw order of all texts and dimensions in the drawing to display in front of all other entities.

Accessing the Command

command bar: *texttofront*
menu bar: *Tools | Draw Order | Bring Text and Dimensions to Front*
toolbar: *Draw Order |*

: ***texttofront***
Prompts you in the command bar:
Bring to front [Text/Dimensions/Both] <Both>: (Press Enter to bring both texts and dimensions to front or choose an option.)

Command Options

Option	Description
Text	Brings text entities only to front.
Dimensions	Brings dimension entities only to front.
Both	Brings both texts and dimensions to front.

Related Commands

DrawOrder - changes the display order of overlapping entities.
DrawOrderByLayer - sets draw order of overlapping objects through layer names.
HatchToBack - sets the draw order of all hatch entities in the drawing to display behind all other entities.

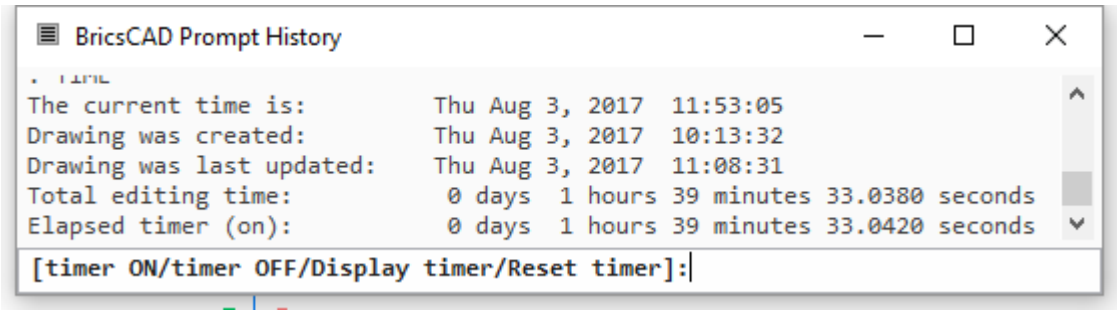
Time

Reports on time in the drawing.

Accessing the Command

command bar: *time*
transparent: *'time*
alias: *ti*
menu bar: *Tools | Time Variables*
toolbar: *Inquiry |*

: ***time***
Prompts you in the Prompt History window:



[timer ON/timer OFF/Display timer/Reset timer]: (Enter an option, or press Enter to exit the command.)

Press the F2 function key to close the Prompt History window.
This command relies on the computer's built-in clock.

Command Options

Option	Description
Timer ON	Turns on the user timer. This can be used to time operations in BricsCAD.
Timer OFF	Turns off the user timer.
Display Timer	Displays the time again. Updates the time.
Reset Timer	Sets the user timer to zero.

Related Command

Status - reports the status of the drawing.

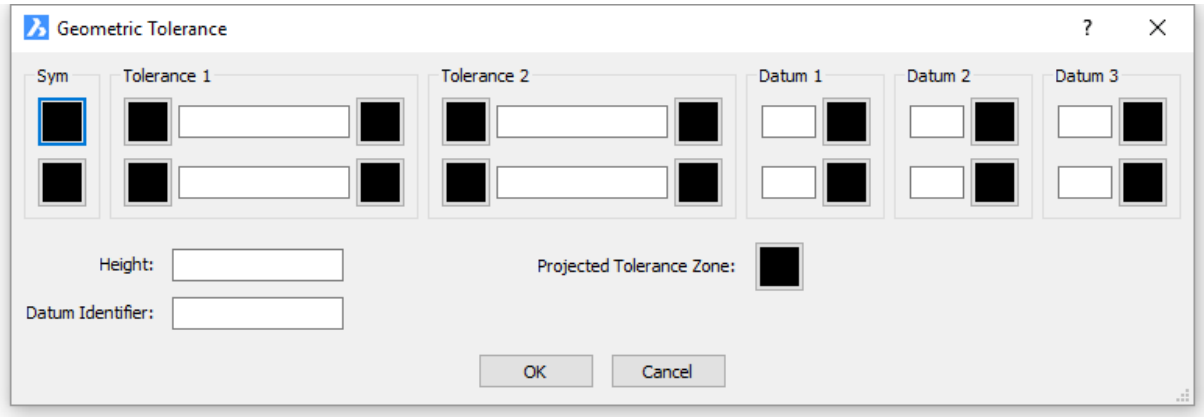
Tolerance

Places tolerance symbols in drawings.

Accessing the Command

command bar: *tolerance*
alias: *tol*
menu bar: *Dimensions | Tolerance*
toolbar: *Dimensions |* ±

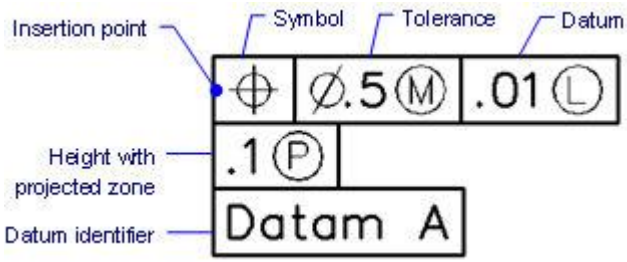
: ***tolerance***
Displays a dialog box:



Select tolerance symbols and enter values, and then click OK.

Prompts you in the command bar:

Select location for tolerance: ***(Specifies the insertion point for the tolerance symbol.)***



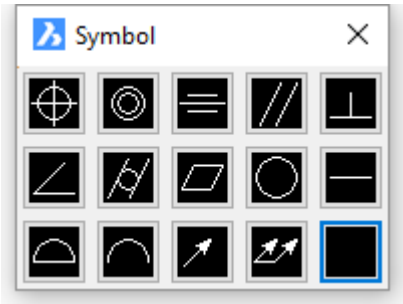
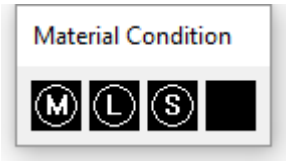
Geometric tolerance symbols

Symbol	Characteristic	Type
	Position	Location
	Concentricity or Coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation
	Cylindricity	Form
	Flatness	Form
	Circularity or Roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular Run-out	Run-out
	Total Run-out	Run-out

Material conditions

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.

Command Options

Option	Description
Symbol	Inserts a tolerance symbol; displays dialog box:  Choose a symbol, and then click OK.
Tolerance 1 Tolerance 2	Specifies tolerance specification: Diameter - toggles the diameter symbol when clicked. Value - specifies the tolerance value. Material condition - displays dialog box when clicked: 
Datum 1 Datum 2 Datum 3	Specifies the datum reference: Value - specifies the datum value. Material condition - click to display the Material Condition dialog box.
Height	Specifies the height of the tolerance symbols.
Datum Identifier	Specifies datum indentifier, such as Datum A.
Projected Tolerance Zone	Toggles the projected tolerance zone symbol:

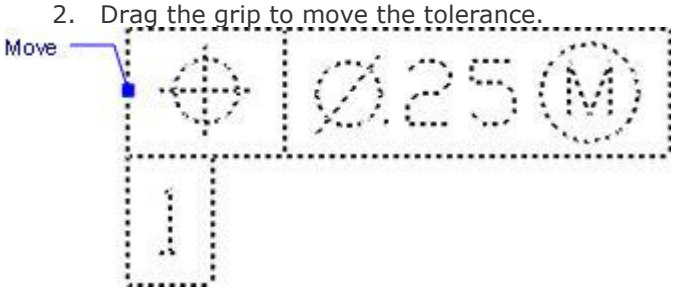
Procedures

To create a geometrical tolerance frame

Grips Editing

Tolerances can be edited directly through grips:

1. Select the tolerance. Notice that it has one grip.
2. Drag the grip to move the tolerance.



Related Commands

Leader - attaches tolerances to leader lines.
DimOrdinate - draws x and y ordinate dimensions.

Toolbar

Toggles the display of toolbars.

Accessing the Command

command bar: *toolbar*

: ***toolbar***
Prompts you in the command bar:
Enter Toolbar name or [All] <All>: (Enter the name of a toolbar, or press Enter for all toolbars.)
Enter an option [Show/Hide] <Show>: (Choose an option.)

Command Options

Option	Description
Enter toolbar name	Specifies the name of a toolbar.
ALL	Specifies all toolbars.
Show	Displays the specified toolbar(s).
Hide	Hides the specified toolbar(s).

Procedures

Toolbars
Toolbar Flyouts
Opening a toolbar
Closing a toolbar
To restore the default toolbars
To set the icon size
To set the position and visibility property of a toolbar

Related Commands

CommandLine - shows the command line.
CommandLineHide - hides the command line.
Customize - customizes toolbars.
CuiLoad and MenuLoad - load CUI files that define toolbars and other user interface elements.
CuiUnload and MenuUnload - load CUI files that define toolbars and other user interface elements.
GraphScr - brings the graphics window to the foreground.
Scrollbar - toggles the display of the scroll bars.
StatBar - toggles the display of the status bar.
TextScr - brings the text window to the foreground.

-Toolbar

Toggles the display of toolbars.

Accessing the Command

command bar: *-toolbar*

: ***-toolbar***

Prompts you in the command bar:
Enter Toolbar name or [All] <All>: (Enter the name of a toolbar, or press Enter for all toolbars.)
Enter an option [Show/Hide] <Show>: (Choose an option.)

Command Options

Option	Description
Enter toolbar name	Specifies the name of a toolbar.
ALL	Specifies all toolbars.
Show	Displays the specified toolbar(s).
Hide	Hides the specified toolbar(s).

Procedures

Toolbars
Toolbar Flyouts
Opening a toolbar
Closing a toolbar
To restore the default toolbars
To set the icon size
To set the position and visibility property of a toolbar

Related Commands

CommandLine - shows the command line.
CommandLineHide - hides the command line.
Customize - customizes toolbars.
CuiLoad and MenuLoad - load CUI files that define toolbars and other user interface elements.
CuiUnload and MenuUnload - load CUI files that define toolbars and other user interface elements.
GraphScr - brings the graphics window to the foreground.
Scrollbar - toggles the display of the scroll bars.
StatBar - toggles the display of the status bar.
TextScr - brings the text window to the foreground.

ToolPalettes

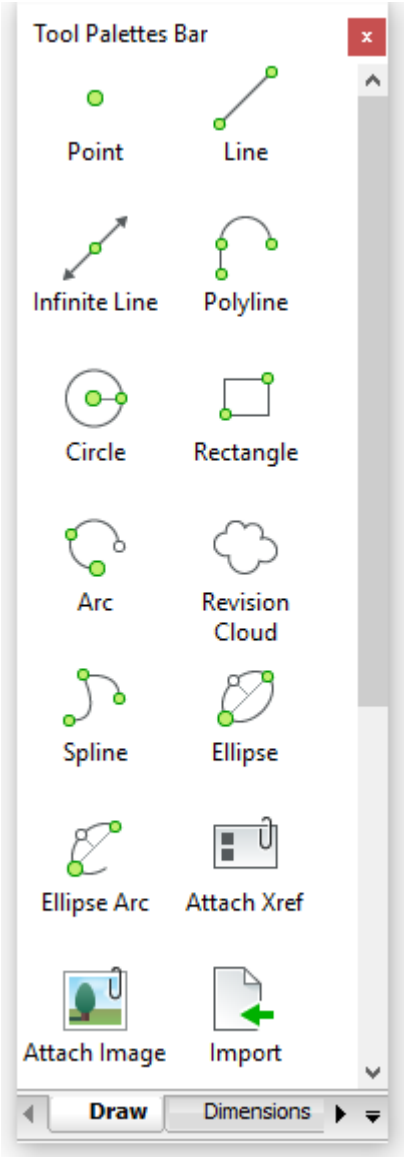
Opens the Tool Palettes panel.

Accessing the Command

command bar: *toolpalettes*
context menu: *Tool Palettes*

: ***toolpalettes***

Displays the Tool Palettes Bar:



Command Options

No options

Procedures

To set the view options
Adding a new tool palette
To open the customize dialog box
Importing a tool palette
Working with tool palette groups
To rename a tool palette
To delete a tool palette
Adding a hatch pattern
Adding a block
Adding a command
Editing a tool
Managing tools
To open or close a panel
To dock a panel
To dock a panel with respect to another docked panel
To stack panels in a tabbed panel

Related Commands

ToolPalettesClose - closes the Tool Palettes panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.
TpNavigate - allows to load a tool palette or tool palette group by name. If not already open, displays the Tool Palettes panel.

ToolPaletttesClose

Closes the Tool Palettes panel.

Accessing the Command

command bar: *toolpalettesclose*

: ***toolpalettesclose***

Command Options

No Options

Related Commands

ToolPalettes - opens the Tool Palettes panel.
-ToolPanel - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

-ToolPanel

Allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Accessing the Command

command bar: *-toolpanel*

: ***-toolpanel***
Prompts you in the command bar:

Enter Tool Panel name <* for all>: (Type the name of a tool panel.)

Enter an option [Show/Hide/Toggle] <Show>: (Choose an option or press Enter to accept the default option.)

Command Options

Option	Description
*	Selects all tool panels. Allows to show/hide/toggle all tool panels: • Mechanical Browser • BIM Libraries • Commandline • Content Browser • Layers • Render Materials • Properties • Report • Ribbon • Sheet Sets • Standard Parts • Structure • Tool Palettes
Show	Displays the tool panel.
Hide	Hides the tool panel.
Toggle	Toggles the display of the tool panel.

Procedures

To open/close a panel

Related Commands

ContentBrowserclose - closes the Content Browser panel.
ContentBrowserOpen - opens the Content Browser panel.
CommandLine - displays the command line.
CommandLineHide - hides the command line.
LayersPanelClose - opens the Layers panel.
LayersPanelOpen - closes the Layers panel.
MatBrowserClose - closes the materials browser.
MatBrowserOpen - opens the materials browser, which shows an overview of available high-resolution render materials.
MechanicalBrowserClose - closes the Mechanical Browser panel.
MechanicalBrowserOpen - opens the Mechanical Browser panel.
StandardPartsPanelclose - closes the Standard Parts panel.
StandardPartsPanelOpen - opens the Standard Parts panel, allowing to insert a standard hardware part as a mechanical component in the current drawing.
Properties - displays the Properties panel with properties of selected entities.
PropertiesClose - closes the Properties panel.
ReportPanelClose - closes the Report panel.
ReportPanelOpen - opens the Report panel.
Ribbon - displays the ribbon.
RibbonClose - hides the ribbon.
SheetSet - Opens the Sheet Sets panel.
SheetSetHide - closes the Sheet Set panel.
ToolPalettes - opens the Tool Palettes panel.
ToolPalettesClose - closes the Tool Palettes panel.

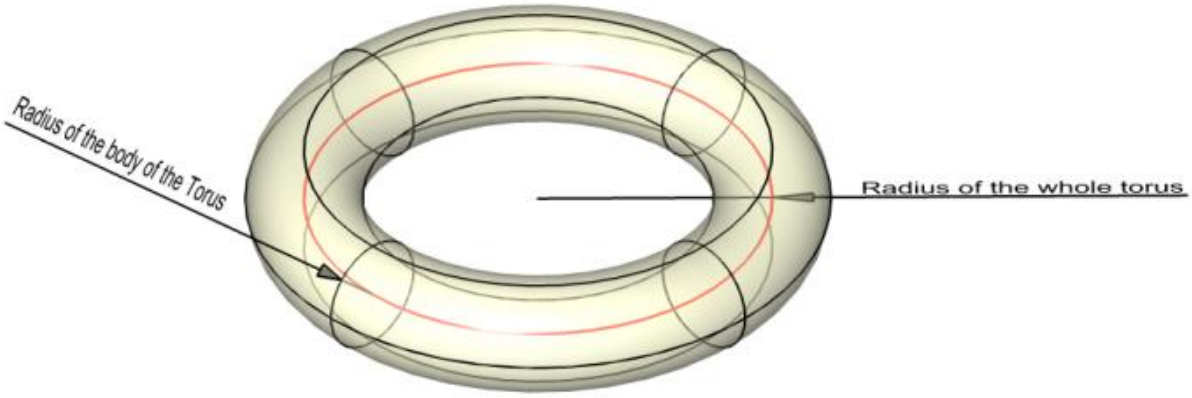
Torus

Creates a torus as a polygon mesh (Classic); creates a torus as a 3D solid (Pro and Platinum).

Accessing the Command

command bar: *torus*
menu bar: *Model| 3D Solids | Torus*
toolbar: *3D Solids |*

: ***torus***
Prompts you in the command bar:
Center of whole torus: <0,0,0> (Pick a point, or enter coordinates.)
Set Radius of whole torus or [Diameter]: (Specify the radius, or type D.)
Set radius of body of torus [Diameter]: (Specify the radius, or type D.)



Command Options

Option	Description
Center of whole torus	Specifies the center of the torus; press Enter to accept the default coordinates.
Radius of whole torus	Specifies the overall radius of the torus; note that the distance is measured to the center of the tube (body).
Radius of body of torus	Specifies the radius of the torus body, or tube. It is not necessary for the body radius to be smaller than the whole radius.
Diameter	Specifies the diameters of the whole torus and/or body.

Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

TpNavigate

Allows to load a tool palette or tool palette group by name. If not already open, displays the Tool Palettes panel.

Accessing the Command

command bar: *tpnavigate*

: ***tpnavigate***
Prompts you in the command bar:

Specify tool palette to display or [palette Group]: (Type the name of a tool palette.)

Command Options

Option	Description
Palette Group	Allows to load a palette group. Prompts you: Specify palette group to display: Type the name of a palette group.

Related Commands

ToolPalettes - opens the Tool Palettes panel.

Trace

Draws wide lines with mitered vertices.

Accessing the Command

command bar: *trace*
menu bar: *Draw | Trace*
toolbar: *Draw | Polygons |*

: ***trace***
Prompts you in the command bar:

Width of trace <0.5000>: (Enter a number, pick two points, or press Enter to accept the default.)
Start of trace: (Pick a point, or enter x,y coordinates.)
Next point: (Pick another point.)
Next point: (Press Enter to exit the command.)

This command draws line segments in an unusual manner.
The trace is not drawn until the second pick point.
The delay allows BricsCAD to determine the mitering needed at vertices.

Three connected trace entities with FILLMODE = ON (left) and FILLMODE = OFF (right).

This command is rarely used today, because the PLine command is more convenient.

Command Options

Option	Description
Width of trace	Specifies the width of the trace. Enter a number or pick two points.
Start of trace	Specifies the start of the trace. Pick a point, or enter x,y coordinates.
Next point	Specifies the subsequent end points. Pick a point, or enter x,y coordinates. Press **Enter** to exit the command. This command miters vertices.

Procedures

Drawing traces

Related Commands

Line - draws line segments.
PLine - draws polyline segments and arcs with constant and variable width.
Solid - draws 3- and 4-sided solid-filled areas.
Donut - draws wide and solid-filled circles.
LWeight - gives most entities visual width.

Transparency

Toggles the transparency of monotone images.

Accessing the Command

command bar: *transparency*
alias: *Alias*
toolbar: *Image* |

: ***transparency***
Prompts you in the command bar:
Enter transparency mode [ON/OFF] <OFF>: (Type ON or OFF.)

When transparency is on, the background color of the image is transparent.

Left: Transparency = OFF, Right: transparency = ON

Command Options

Option	Description
ON	Turns on transparency.
OFF	Turns off transparency.

Related Commands

Image - controls the parameters of the image file.
ImageAttach - places raster images in drawings.
-ImageAttach - attaches raster images to the drawing; displays prompts in the command bar.
ImageAdjust - changes the properties of images through the Properties palette.
ImageQuality - toggles the quality of the image.

Trim

Trims entities to a cutting line.

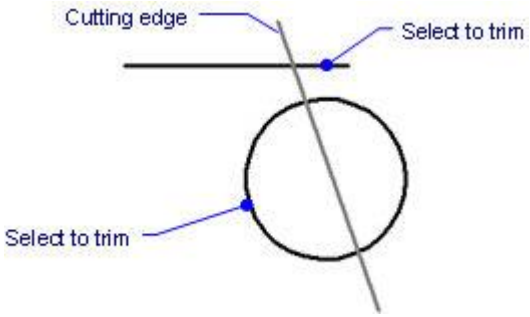
Accessing the Command

command bar: *trim*
alias: *tr*
menu bar: *Modify | Trim*
toolbar: *Modify |*

: ***trim***
Prompts you in the command bar:

Select cutting entities for trim <ENTER to select all>: (Choose one or more entities, or press Enter to select all entities in the drawing.)
Entities in set: 1

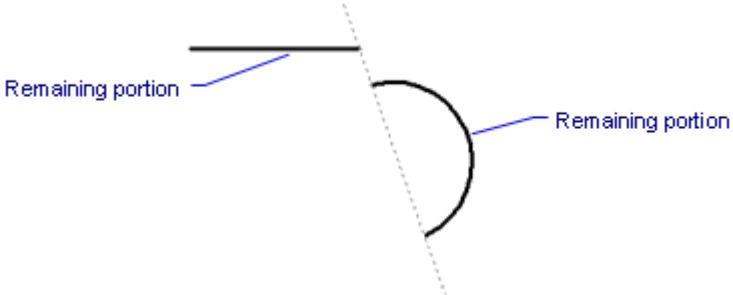
Select cutting entities for trim <ENTER to select all>: (Press Enter to continue the command.)



Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase]: (Pick an entity or enter an option or press shift, then pick an entity to extend.)

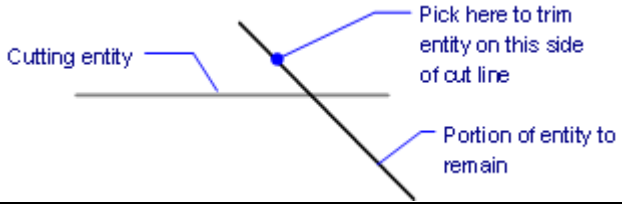
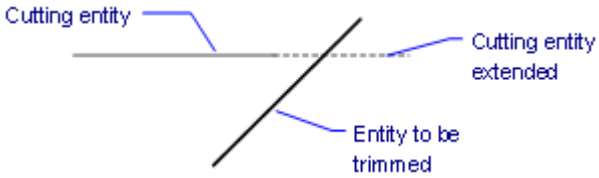
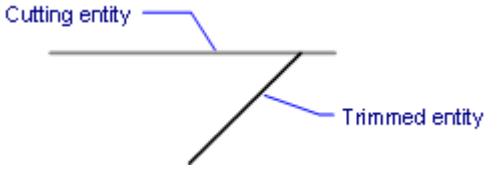
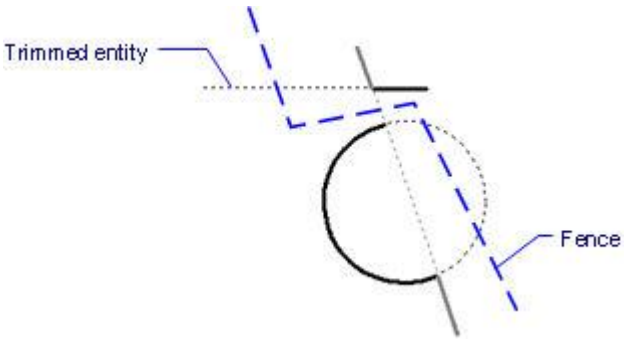
Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase/Undo]: (Pick another entity or press shift, then pick an entity to extend.)

Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase/Undo]: (Pick another entity, or press shift, then pick an entity to extend, or press Enter to stop .)



Command Options

Option	Description
Shift key	Extends the picked entities instead of trimming them. When executing the Trim or the Extend command, pressing the shift key switches between trimming and extending.
Select cutting entities for trim	Choose one or more entities to act as cut lines, including text. These entities define where crossing entities are trimmed.

	Select one or more entities using any selection method.
ENTER to select all	Selects all entities in the drawing, except for those on frozen layers. Press Enter to select all. Note that entities can both define cutlines and be cut.
Select entity to trim	Chooses the entity to be trimmed. Select a single entity; only one entity can be trimmed at a time. Your pick point defines the portion of the entity to be removed. 
Edge mode	Toggles edge mode; prompts you: Extend/No extend - type E or N. • Extend - trims entities that do not physically intersect the cutting entity.  • No extend - trim only entities that intersect.  See Projection option.
Fence	Cuts entities interactively; prompts you: First point of fence - specify the fence's starting point. Next fence point - specify the next point; press Enter to exit this option. As you pick fence points in the drawing, BricsCAD immediately trims entities under the fence. The portion of each entity between other entities is trimmed:  Use this option together with ALL (select all entities to trim).

Crossing	Cuts entities interactively; prompts you: First corner of crossing window - specify the first corner Second corner of crossing window - specify the second corner. The portions of the entities that cross the specified rectangle are trimmed.
Projection	Defines how intersections are projected; prompts you: None/Ucs/View - enter an option. This option affects the operation of the Edge option.
None	Intersections are not projected. Only those entities that intersect are trimmed.
UCS	Intersections are projected into the x,y plane of the current UCS. Entities that appear to intersect are trimmed.
View	Intersections are projected along the current viewpoint. Entities that appear to intersect are trimmed.
Erase	Deletes selected entities; prompts you: Select entities to delete - select the entities
Undo	Undoes the last trim action. Repeat this option undo earlier trim actions.

Procedures

To trim entities

Related Commands

Break - cuts parts out of entities.
Chamfer - chamfers intersections.
Extend - Extends open entities to a boundary.
Fillet - fillets intersections.
Settings - specifies parameters for trims.

TxtExp

Explodes text into polylines (short for "text explode").

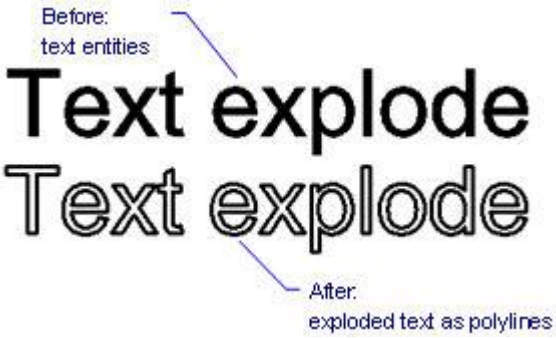
Accessing the Command

command bar: *txtexp*
menu: *Modify | Explode Text*

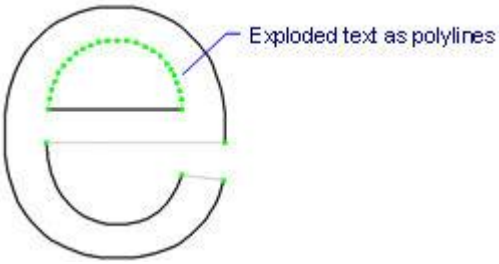
: ***txtexp***
Prompts you in the command bar:

Select text to be exploded: (Choose one or more text entities.)

Select text to be exploded: (Choose more text entities, or press Enter to end entity selection.)



SHP and TTF fonts are exploded to individual polylines and polyarcs.



Command Options

Option	Description
Select text to be exploded	Chooses the text to be exploded into polylines. Use any entity selection method. Press Enter to end entity selection.

Procedures

To explode text

Related Commands

Text and -Text - create text.
MText and -MText - create paragraph text.
Explode - explodes non-text entities.
U - undoes the explosion.

U

U

Undoes the last command.

Accessing the Command

command bar: *u*
shortcut: *Ctrl+Z*
menu bar: *Edit | Undo*
toolbar: *Standard |*

: ***u***
Reports to you in the command bar:
Undo: (name of command)

Command Options

No options.

Related Commands

Undo - undoes one or more commands.
Redo - redoes the last undone command.
Oops - unerases the last erased object, including newly created blocks.
Zoom - utilizes the Previous option to show the previous view.

Ucs

Creates and displays UCSs through the command bar (short for "user-defined coordinate system").

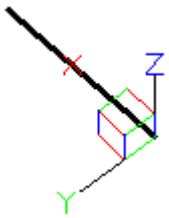
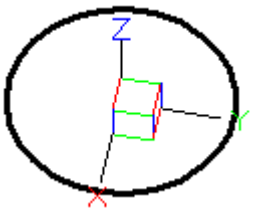
Accessing the Command

command bar: *ucs*

: ***ucs***
Prompts you in the command bar:
Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World]
<World>: (Pick a point to specify the origin, enter an option, or press Enter for the WCS.)

Command Options

Option	Description
World	Switches to the WCS (world coordinate system).
Specify origin of UCS	Specifies a UCS through points picked in the drawing. Pick a point, or enter x,y,z coordinates; prompts you: Point on X-axis/<Accept> - enter an option. • Point: Defines the X-axis. • Accept: Defines the UCS using a new origin point and keeping the orientation of the axes. Point on the XY-plane with positive Y value/<Accept> - enter an option. • Point: Defines the XY-plane and the direction of the Y-axis and Z-axis. • Accept: Defines the UCS using the new X-axis, keeping the XY plane and the direction of the Z-axis.
Face	Aligns the UCS to a face of a 3D solid; prompts you: Select entity - click the face of a 3D solid. The X-axis of the UCS lies on the edge by which the cursor enters the face. Enter an option [Flip/Xrotate/Yrotate] <accept> - choose an option or press Enter to accept the UCS. • 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 named UCSs; prompts you: Enter an option [Restore/Save/Delete/? for ucs list]: - enter an option. • Restore: Restores a named UCS; prompts you: UCS name to restore - enter the name of a UCS. As a shortcut, you can enter R directly at the 'Specify origin of UCS...' prompt. • Save: Saves the current UCS by a name; prompts you: Enter name for saving UCS, or ? for list - specifies the name to give the UCS. As a shortcut, you can enter S directly at the 'Specify origin of UCS...' prompt. • Delete: Erases a named UCS from the drawing; prompts you: UCS name(s) to delete, or ? for list - enter one or more UCS names. Separate multiple names with commas. As a shortcut, you can enter D directly at the 'Specify origin of UCS...' prompt.

	• ? for ucs list: Lists the names of UCSs in the drawing; prompts you: Name(s) of UCS to list <* for all> - specify the names to list: * - lists all UCS names. name* - lists names of UCSs that begin with name. name - lists the UCS specified by name. If necessary, press **F2** to view the history prompt window. As a shortcut, you can enter ? directly at the 'Specify origin of UCS...' prompt.
Entity	Aligns the UCS to a selected entity; prompts you: Select entity for UCS definition - choose a single entity. The orientation and type of entity define the orientation of the UCS.  • UCS origin - located to the end point, center point, or vertex closest to the pick point. • X axis - aligned with the entity r an edge. • XY plane - aligned to the entity's plane. • Orientation - maintained for ambiguous objects. 
Previous	Changes the UCS to the previous one.
View	Sets the UCS to the current viewpoint. The x axis and y axis are parallel to the view edges and the z axis is perpendicular to the view, positive Z-axis pointing to the viewer. The origin is copied from the previous coordinate system.
X	Rotates the current UCS about the x axis; prompts you: Enter angle of rotation about X axis - specify an angle or pick two points.
Y	Rotates the current UCS about the y axis; prompts you: Enter angle of rotation about Y axis - specify an angle or pick two points.
Z	Rotates the current UCS about the z axis; prompts you: Enter angle of rotation about Z axis - specify an angle or pick two points.
ZAxis	Sets the UCS relative to the z-axis; prompts you to define the Z-axis: Origin point - pick a point, or enter x,y,z coordinates. Point on positive Z-axis - pick another point, or enter x,y,z coordinates. The second point defines the positive Z-axis. The XY-plane is perpendicular to the Z-axis with the X-axis horizontal and the Y-axis pointing upwards.

Move	Moves the origin of the current UCS, keeping the orientation of the axes; prompts you: Origin point <0,0,0> - pick point, or enter x,y,z coordinates.

Related Commands

ExpUcs - creates, modifies, and deletes named UCSs through the Drawing Explorer.
UcsIcon - controls the appearance of the UCS icon.
SetUcs - sets the UCS to a specified viewpoint.
Plan - changes the viewpoint to the plan view of the WCS or current UCS.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
Settings - sets parameters of UCSs.

UcsIcon

Controls the display of the UCS icon (short for "user-defined coordinate system icon").

Accessing the Command

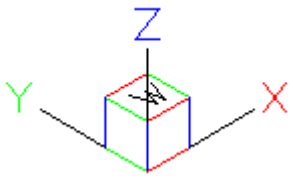
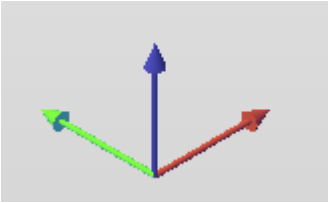
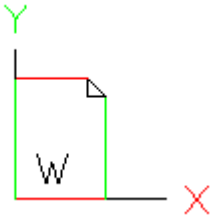
command bar: *ucsicon*

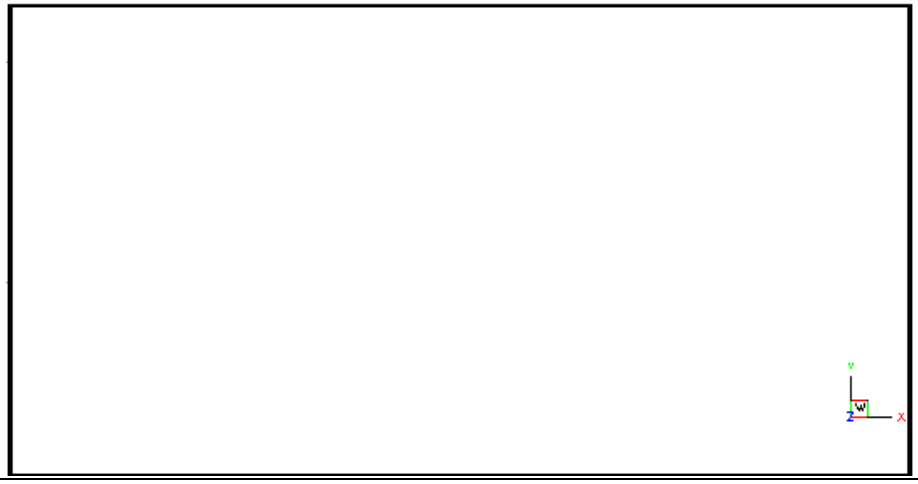
: ***ucsicon***
Prompts you in the command bar:

[ucs icon ON/ucs icon OFf/display in All views/display at ORigin/display in Corner] <ON>: (Enter an option.)

You can also change the properties of the UCS icon in the User Coordinate System section of the Settings dialog box.

Command Options

Option	Description
ON	Turns on the display of the UCS icon.  UCS icon in Wireframe or Hidden visual style.  UCS icon in a rendered visual style.  UCS icon in a paperspace layout. The color of the axes is controlled by the COLORX, COLORY and COLORZ system variables. X axis color: Red Y axis color: Green Z axis color: Blue

OFF	Turns off the UCS icon.
All	Applies changes to all viewports; repeats prompts: [ucs icon ON/ucs icon OFf/display at ORigin/display in Corner] <ON> - enter an option to apply to all viewports. This command normally applies only to the current viewport; use this option to apply changes to all viewports.
ORigin	Displays the UCS icon at the origin (0,0). When the origin is outside of the viewport, the UCS icon is displayed at the corner defined by the UCSICONPOS system variable (see Corner below) 
Corner	Displays the UCS icon in the viewport corner defined by the UCSICONPOS system variable: • 0: Lower right corner • 1: Lower left corner • 2: Upper right corner • 3: Upper left corner

Related Commands

Ucs - creates and modifies user-defined coordinate systems.
ExpUcs - creates, modifies, and deletes named UCSs through the Drawing Explorer.
SetUcs - sets the UCS to a specified viewpoint.
Plan - changes the viewpoint to the plan view of the WCS or current UCS.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
Settings - sets parameters of UCSs.

Undefine

Removes commands from BricsCAD, temporarily.

Accessing the Command

command bar: *undefine*

: ***undefine***
Prompts you in the command bar:
Command to undefine: (Enter the name of a command.)

When you enter the name of an undefined command, BricsCAD responds:
Unable to recognize command. Please try again.

Use the Redefine command to recover the command.
Undefined commands can be launched when preceded with a period (.CommandName).

Command Options

Option	Description
Command to undefine	Name a command to remove.

Related Commands

Commands - lists the names of commands.
Customize - creates new commands through macros and LISP routines.
Redefine - activates commands that were deactivated with the Undefine command

Undo

Undoes one or more commands; some commands cannot be undone.

Accessing the Command

command bar: *undo*
toolbar: *Standard* |

: ***undo***
Prompts you in the command bar:

Undo: Number of steps to undo or [Mark/Back to mark/BEgin set/End set/Control/Auto]: (Enter the number of commands to undo, or enter an option.)

BricsCAD reports the commands being undone. For example, enter the Customize and then the Units commands. Enter Undo:

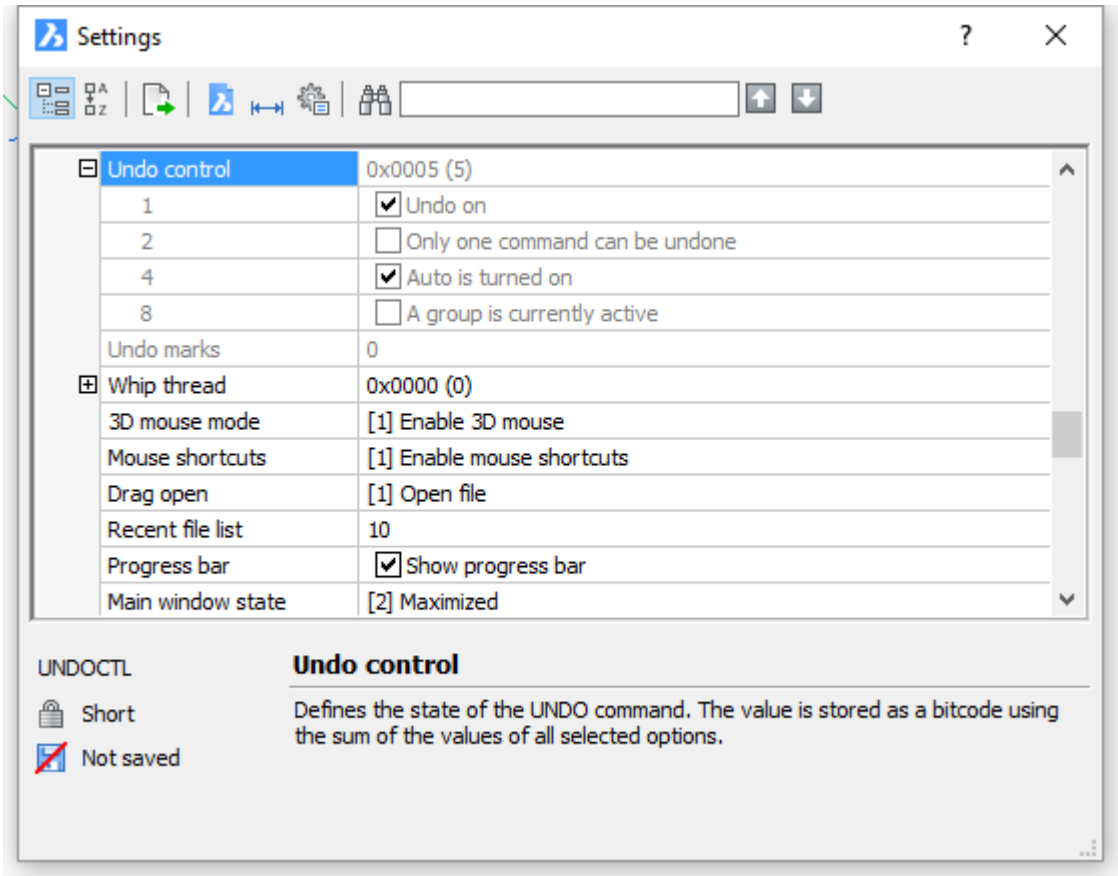
Undo: (UNITS)

Undo: (CUSTOMIZE)

Command Options

Option	Description
Number of steps to undo	Specifies the number of commands to undo. Enter a number. Commands are undone sequentially; you cannot skip over commands. Some commands cannot be undone, such as Save, Open, and Plot.
Mark	Sets a marker. The Back to mark option undoes all commands to the marker.
Back to mark	Undoes all commands back to when the marker was set.
BEgin set	Sets the start to a group that groups the following commands. The Undo command treats the commands in the set as a single undo.
End set	Ends the group.
Control	Specifies options; prompts you: Set Undo control: None/One/<All> - enter an option. • None: Turns off the undo mechanism. This is useful when disk space is running low, because the undo mechanism uses disk space. • One: Limits this command to a single undo. This option turns the Undo command into the **U** command. • All: Turns on the undo mechanism.
Auto	Considers all commands executed by a macro as a single undo.

The Undo control settings are stored in the UNDOCTL system variable, which is saved as a bitcode using the sum of the values of all selected options.
In BricsCAD four options are available:



By default UNDOCTL=5. Since the value of UNDOCTL is not saved, the value is reset to the default with each BricsCAD session.

In AutoCAD two more options exist:

- 16: Zoom and pan operations are grouped as a single action
- 32: Layer property operations are grouped as a single action

BricsCAD behaves as if those options are ON.

Related Commands

U - undoes the last command.
Redo - redoes the last undone command.
Oops - unerases the last erased object, including newly created blocks.
Zoom - utilizes the Previous option to show the previous view.

Union

Performs Boolean union operations on 3D solids and 2D regions; adds one to another to form a single entity.

Accessing the Command

command bar: *union*
alias: *uni*
menu bar: *Model | 3D Solid Editing | Union*
toolbar: *3D Solid Editing |*

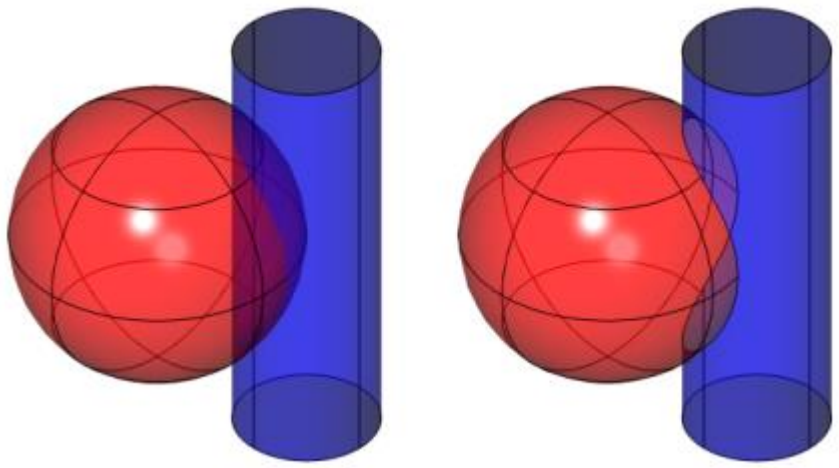
: ***union***
Prompts you in the command bar:

Select ACIS objects to union: (Choose one or more 3D solids and/or 2D regions.)

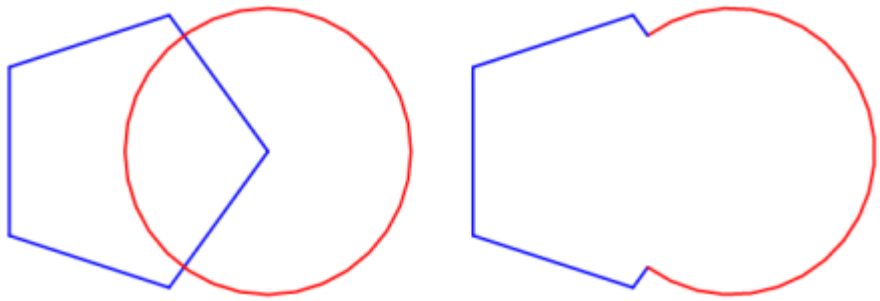
Entities in set: 2

Select ACIS objects to union: (Choose more ACIS entities, or press Enter to complete the command.)

3D solids that are joined through the Union command often look no different after the operation:



This command also applies to regions:



Command Options

Option	Description
Select ACIS objects to union	Specifies the ACIS entities to join into a single entity. Choose one or more 3D solids and/or 2D regions. The resulting entity takes on the properties of the ACIS entity selected first. This command also joins regions and 3D solids into a single entity.

Related Commands

BimSplit - splits segments of wall-like solids into separated solids.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Subtract - removes one solid model from a second.

UnisolateObjects

Unhides entities that were hidden.

Accessing the Command

command bar: *unisolateobjects*
aliases: *unisolate / unhideobjects / unhide*
toolbar: *Hide/Show Entities* |
Quad: *Custom* |

: ***unisolateobjects***

Command Options

No options.

Related Commands

HideObjects - hides the selected entities.
IsolateObjects - hides all but the selected entities.

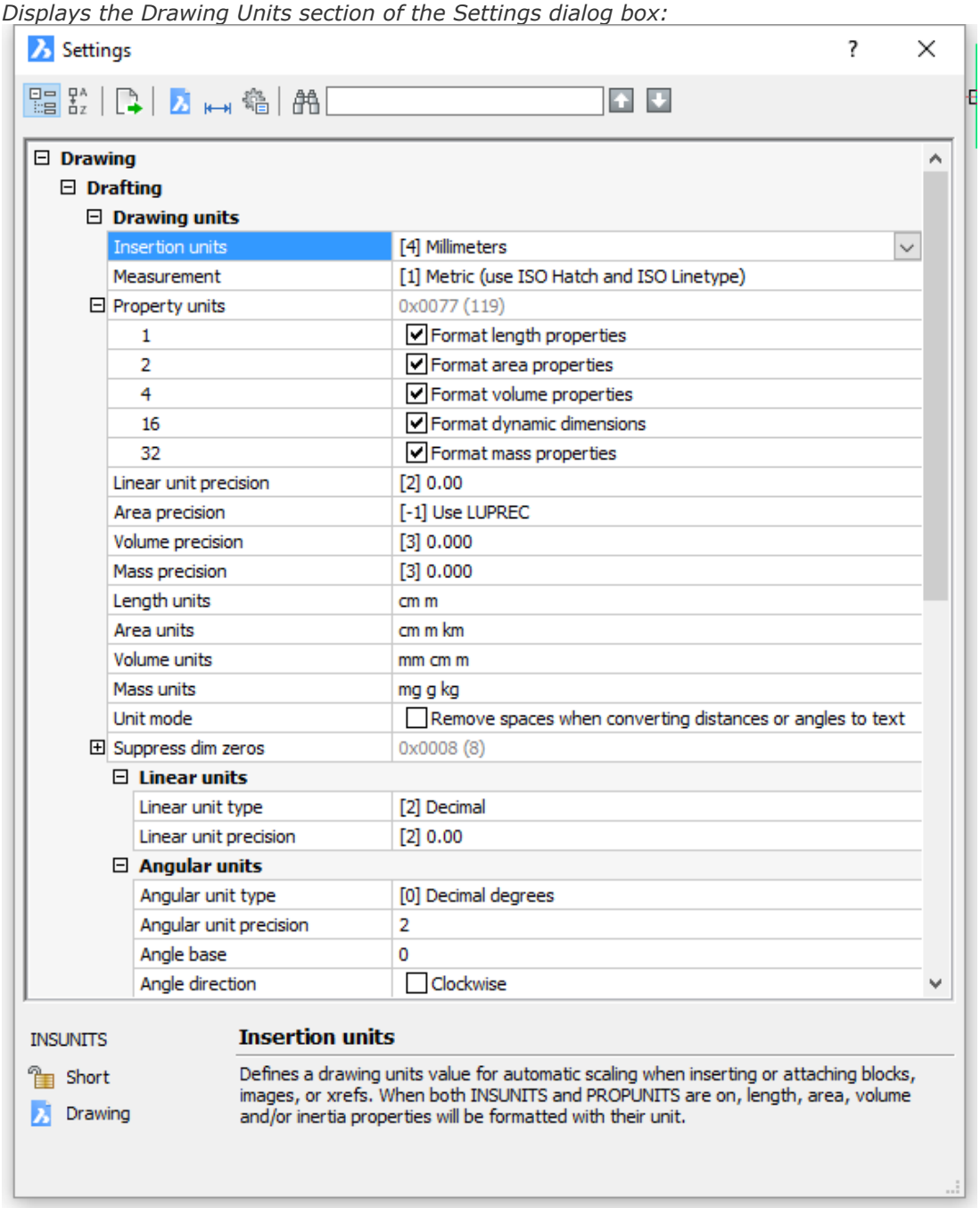
Units

Sets units of linear and angular measurements through the Settings dialog box.

Accessing the Command

command bar: *units*
transparent: *'units*
alias: *un, ddunits*
status bar: *X,y,z-coordinates*

: ***units***
Displays the Drawing Units section of the Settings dialog box:

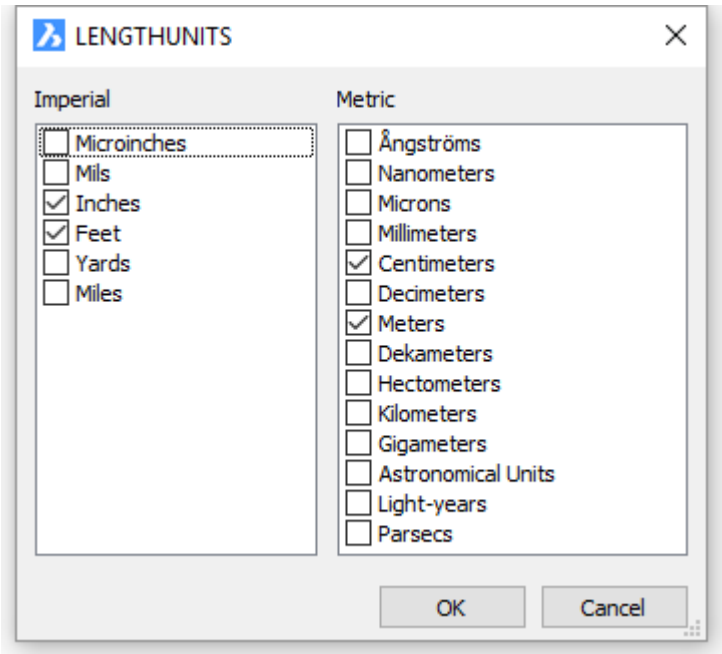
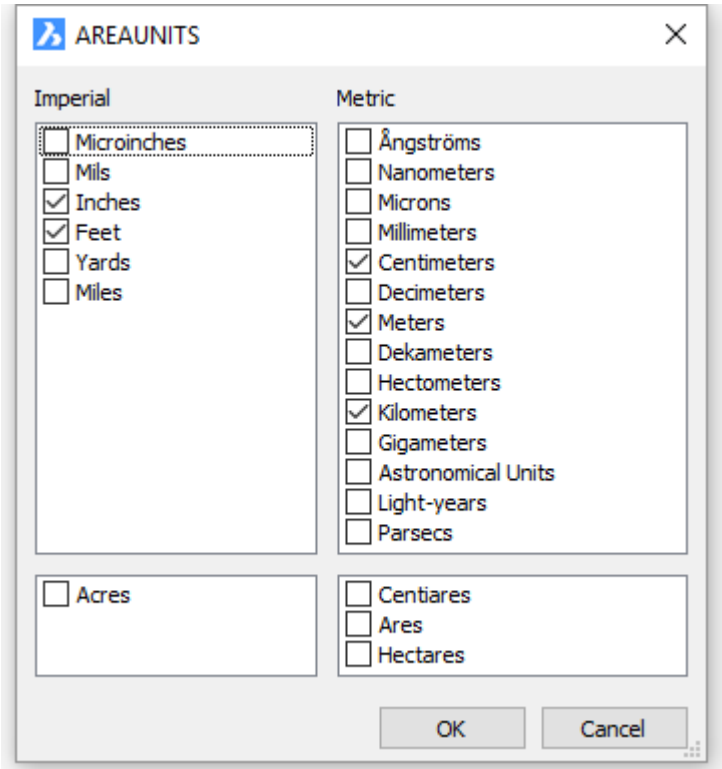


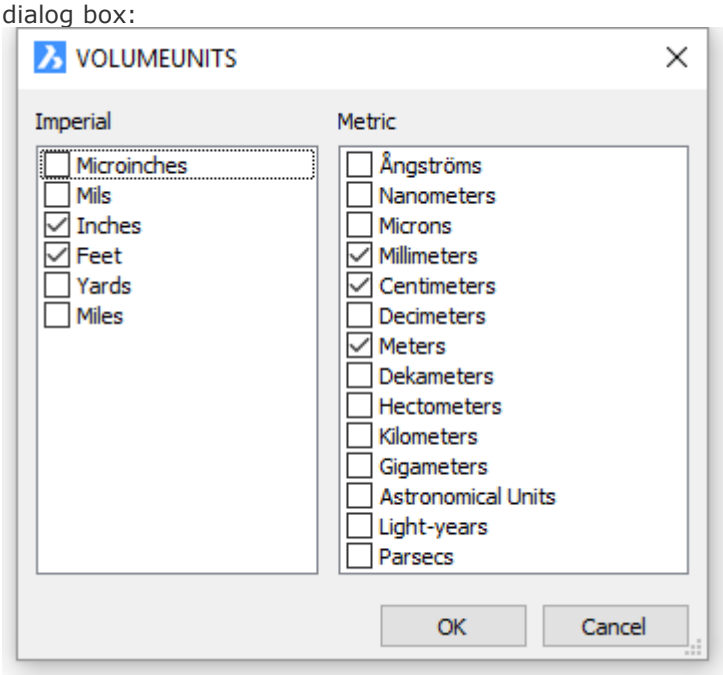
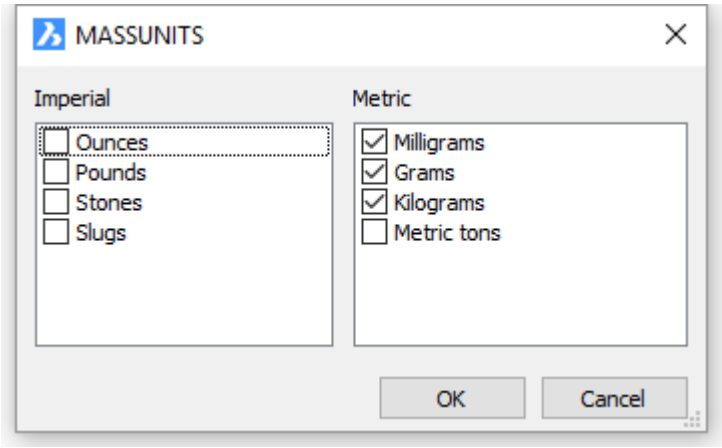
Change settings, and then click the X button.

Command Options

Option	Description
Insertion Units	Defines a real world value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs. When both INSUNITS and PROPUNITS are ON, Length, Area, Volume and/or Inertia properties will be formatted with a dedicated unit. Select a unit: [0] Unspecified (No units) [1] Inches [2] Feet [3] Miles [4] Millimeters [5] Centimeters [6] Meters [7] Kilometers [8] Microinches [9] Mils [10] Yards [11] Angstroms [12] Nanometers [13] Microns [14] Decimeters [15] Decameters [16] Hectometers [17] Gigameters [18] Astronomical Units [19] Light Years [20] Parsecs
Measurement	Defines which default source files will be used when loading linetypes and predefined hatch patterns (see the Hatch command). Select an option: [0] Imperial (use ANSI Hatch and ANSI Linetype) [1] Metric (use ISO Hatch and ISO Linetype) • Imperial: uses default.lin and default.pat • Metric: uses iso.lin and iso.pat BricsCAD searches for the default hatch and linetypes files in the folders which are defined by the SRCHPATH system variable. Default support file search path: C:\Users\<username>\AppData\Roaming\Bricsys\Bricscad\V..\en_US\Support
Property Units	Defines whether certain types of property values are automatically formatted when INSUNITS specified (not zero). The units to be used are defined by the corresponding units variable: Length, Area, Volume or Mass. The value is stored as a bitcode, using the sum of the values of all selected options: 1 = format length properties 2 = format area properties 4 = format volume properties 16 = format dynamic dimensions

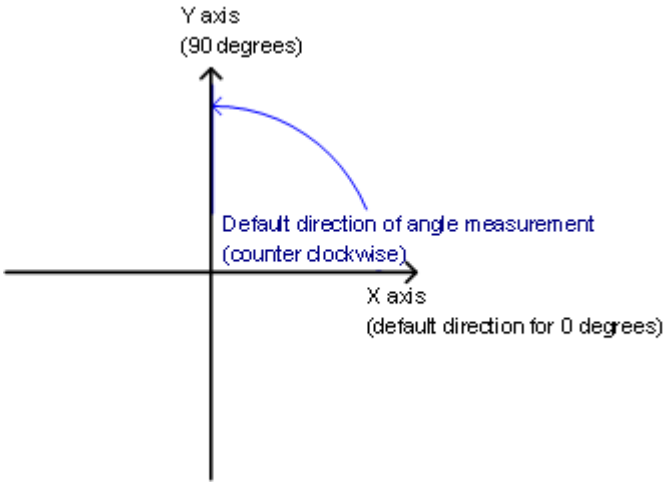
	32 = format mass properties
Linear unit precision	Specifies the number of decimal places. Click the field, then select a value in the list: [0] 0 [1] 0.0 [2] 0.00 [3] 0.000 [4] 0.0000 [5] 0.00000 [6] 0.000000 [7] 0.0000000 [8] 0.00000000 This option affects the precision of fractions in architectural 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 The value of Linear unit precision (LUCPREC) applies also to AREAPREC, VOLUMEPREC and MASSPREC unless these system variables are specified explicitely.
Area Precision Volume Precision Mass Precision	Specifies the number of decimal places for Areas, Volumes an Masses when the corresponding bit of the PROPUNITS (Property Units) system variable is ON. If set negative, the value of LUPREC (Linear Unit Precision) is used. Click the settings field, then select a value in the list: [-1] Use LUPREC [0] 0 [1] 0.0 [2] 0.00 [3] 0.000 [4] 0.0000 [5] 0.00000 [6] 0.000000 [7] 0.0000000 [8] 0.00000000
Length Units	Defines a list of units used for displaying lengths when the corresponding bit of PROPUNITS is ON. If empty, all quantities display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used. Click the settings field, then select the desired units in the LENGTHUNITS dialog box:

	
Area Units	Defines a list of units used for displaying areas when the corresponding bit of PROPUNITS is ON. If empty, all quantities display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used. Click the settings field, then select the desired units in the AREAUNITS dialog box: 
Volume Units	Defines a list of units used for displaying volumes when the corresponding bit of PROPUNITS is ON.

	If empty, all quantities display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used. Click the settings field, then select the desired units in the VOLUMEUNITS dialog box: 
Mass Units	Defines a list of units used for displaying masses when the corresponding bit of PROPUNITS is ON. If empty, all quantities display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used. Click the settings field, then select the desired units in the MASSUNITS dialog box: 
Unit mode	Specifies how imperial units display. 0 = Displays fractional, feet-and-inches, and surveyor's angles in "report" format using spaces as delimiters. 1 = Displays fractional, feet-and-inches, and surveyor's angles in "input" format without including spaces and, in some cases, substituting dashes for spaces

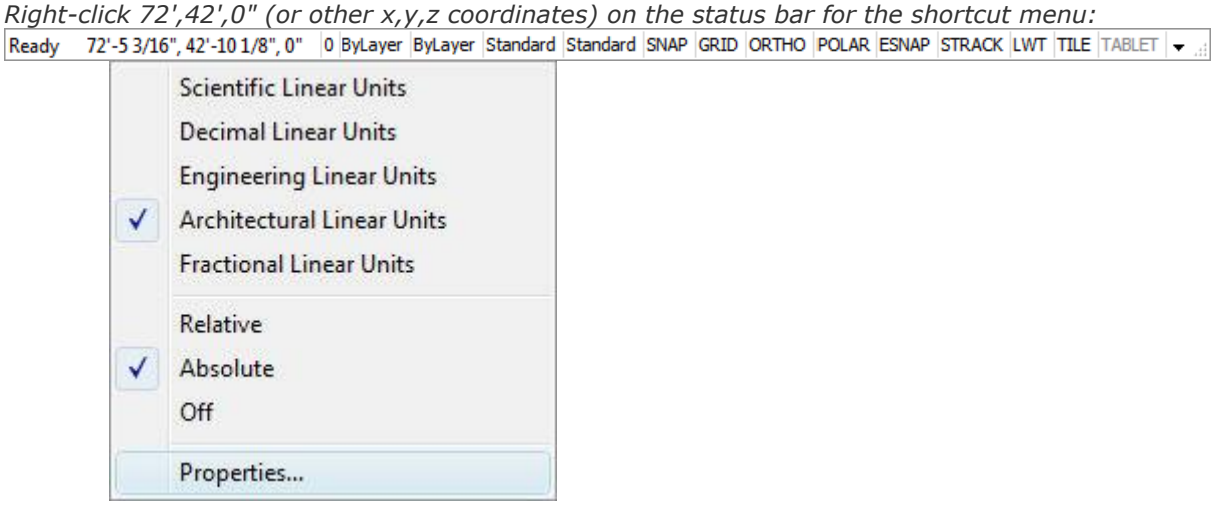
<table>
<tr><td>Suppress dim zeros</td><td>Allows to suppress leading and / or trailing zeros in decimal dimensions.
Allows to suppress zero feet and zero inches in imperial dimensions.

<table><tr><td>Suppress dim zeros</td><td>0x0000 (0)</td></tr><tr><td>mask: 0x0003</td><td>[0] Suppress zero feet and precisely zero inches</td></tr><tr><td>4</td><td>☐ Suppress leading zeros in decimal dimensions</td></tr><tr><td>8</td><td>☐ Suppress trailing zeros in decimal dimensions</td></tr></table></td></tr>
<tr><td>Linear unit type</td><td>Specifies the units of linear measurement. Select a unit:
[1] Scientific
[2] Decimal
[3] Engineering
[4] Architectural
[5] Fractional
Scientific - such as 4.225E+01
Decimal - such as 42.25
Engineering - such as 3'-6.25"
Architectural - such as 3'-6 1/4"
Fractional - such as 42 1/4
For metric units, choose 2 (decimal).</td></tr>
<tr><td></td><td></td></tr>
<tr><td>Angular unit type</td><td>Specifies the type of angular units. Select a unit:
[0] Decimal degrees
[1] Degrees/minutes/seconds
[2] Gradians
[3] Radians
[4] Surveyor's units
Decimal degrees - such as 90.0.
Degrees/minutes/seconds - such as 90d0'0.
Gradians - such as 100.00g. There are 400 grads in a circle.
Radians - such as 1.57r. There are 2*pi radians (approximately 6.282) in a circle.
Surveyor's units - such as N 00d0'0"E. The N and E in Surveyor's units refers to North and east.</td></tr>
<tr><td>Angular unit precision</td><td>Specifies the precision for angular units.
Click the field, then select a value in the list:
[0] 0
[1] 0.0
[2] 0.00
[3] 0.000
[4] 0.0000
[5] 0.00000
[6] 0.000000
[7] 0.0000000
[8] 0.00000000</td></tr>
</table>

Angle base	Specifies the direction for 0 degrees. The default is the positive x axis. Enter an angle. You can also rotate the drawing through the **Snap** command's Rotate option. 
Angle direction	Toggles the direction in which angles are measured: • No - measures angles counterclockwise (default). • Yes - measures angles clockwise.

Status Bar Menu

Right-click 72',42',0" (or other x,y,z coordinates) on the status bar for the shortcut menu:



Shortcut Menu	Description
Scientific Linear Units	Chooses scientific as the working units.
Decimal Linear Units	Chooses decimal (metric) as the working units.
Engineering Linear Units	Chooses engineering as the working units.
Architectural Linear Units	Chooses architectural as the working units.
Fractional Linear Units	Chooses fractional as the working units.
Relative	Displays relative coordinates in the status bar.
Absolute	Displays absolute coordinates in the status bar.
Off	Displays coordinates only when points are picked.
Properties	Opens the Coordinates section of the Settings dialog box. ⊟ **Drawing** ⊟ **Drafting** ⊞ **Drawing units** ⊞ **Coordinate input** ⊟ **Display/Viewing** Coordinates | Coordinates always show pointer location Update coordinates only when selecting points Coordinates always show pointer location Coordinates in polar form for point, distance and angle selection • Update coordinates only when selecting points - equivalent to Off. • Coordinates always show pointer location - equivalent to Absolute. • Coordinates in polar form for point, distance and angle selection - equivalent to Relative.

Related Commands

-Units - sets units through the command bar.
NewWiz - sets units and other parameters for new drawings.
New - starts new drawing in Imperial or metric units.

-Units

Sets units of linear and angular measurements in the command bar.

Accessing the Command

command bar: *-units*
transparent: *'-units*
alias: *-un*
status bar: *x,y,z-coordinates*

: ***-units***
Prompts you in the command bar:

Unit mode (LUNITS):

1. Scientific 4.225E+01
2. Decimal 42.25
3. Engineering 3'-6.25"
4. Architectural 3'-6 1/4"
5. Fractional 42 1/4

Unit mode 1-5 <2>: (Enter a number.)

Number of decimal places for LUNITS 0-8 <4>: (Enter a number.)

Angular unit mode (AUNITS):

1. Decimal degrees 90.0
2. Degrees/minutes/seconds 90d0'0
3. Grads 100.00g
4. Radians 1.57r
5. Surveyor's units N 00d0'0"E

Angular unit mode 1-5 <1>: (Enter a number.)

Number of decimal places for angular units 0-8 <0>: (Enter a number.)

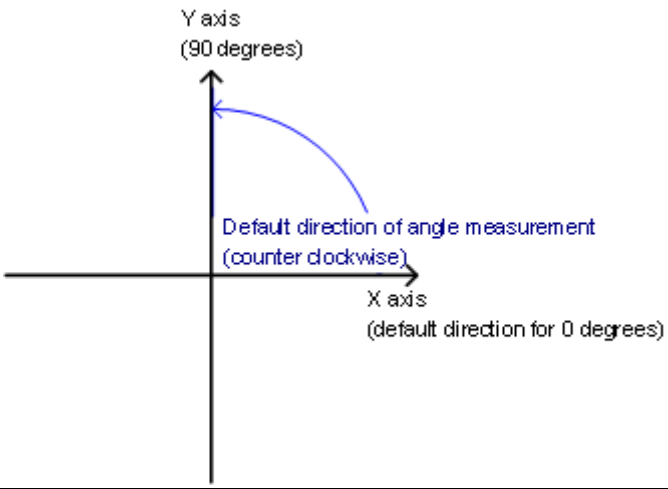
Angle 0 direction <0>: (Enter an angle.)

Do you want angles measured clockwise? <N>: (Type Y or N.)

Command Options

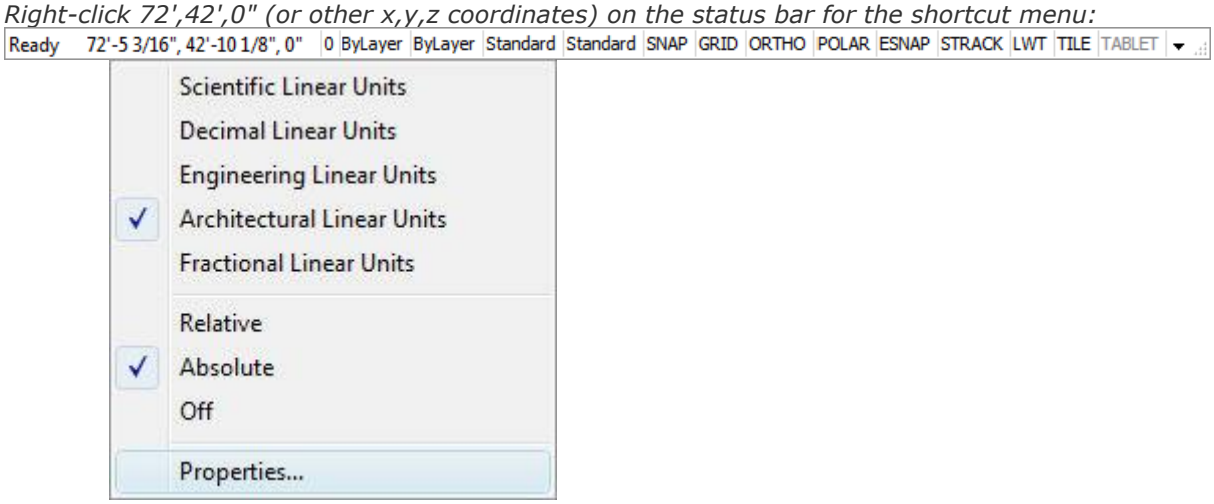
Option	Description
Unit mode	Specifies the units of linear measurement. Enter a number between 1 and 5 corresponding to the list: • **1 -** Scientific, such as 4.225E+01 • 2 - Decimal, such as 42.25 • 3 - Engineering, such as 3'-6.25" • 4 - Architectural, such as 3'-6 1/4" • 5 - Fractional, such as 42 1/4 For metric units, choose 2 (decimal). (LUNITS refers to the system variable that stores the type of linear units.)
Number of decimal places for LUNITS	Specifies the number of decimal places. 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 affects the precision of fractions in architectural and fractional units: • 0 - 0 fractional precision, such as 1 • 1 - 1/2 • 2 - 1/4 • 3 - 1/8 • 4 - 1/16 • 5 - 1/32 • 6 - 1/64 • 7 - 1/128 • 8 - 1/256
Angular unit mode	Specifies the type of angular units. Enter a number between 1 and 5: • 1 - Decimal degrees, such as 90.0 • 2 - Degrees, minutes, and decimal seconds, such as 90d0'0 • 3 - Grads, such as 100.00g • 4 - Radians, such as 1.57r • 5 - Surveyor's units, such as N 00d0'0"E There are 400 grads in a circle. There are 2*pi radians (approximately 6.282) in a circle. The N and E in Surveyor's units refers to North and east. (AUNITS refers to the system variable that stores the type 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).

Status Bar Menu

Right-click 72',42',0" (or other x,y,z coordinates) on the status bar for the shortcut menu:



Shortcut Menu	Description
Scientific Linear Units	Chooses scientific as the working units.
Decimal Linear Units	Chooses decimal (metric) as the working units.
Engineering Linear Units	Chooses engineering as the working units.
Architectural Linear Units	Chooses architectural as the working units.
Fractional Linear Units	Chooses fractional as the working units.
Relative	Displays relative coordinates in the status bar.
Absolute	Displays absolute coordinates in the status bar.
Off	Displays coordinates only when points are picked.
Properties	Opens the Coordinates section of the Settings dialog box. ⊟ **Drawing** ⊟ **Drafting** ⊞ **Drawing units** ⊞ **Coordinate input** ⊟ **Display/Viewing** Coordinates: Coordinates always show pointer location Update coordinates only when selecting points Coordinates always show pointer location Coordinates in polar form for point, distance and angle selection
Coordinates	Specifies how coordinates are displayed in the status bar: • Update coordinates only when selecting points - equivalent to Off. • Coordinates always show pointer location - equivalent to Absolute. • Coordinates in polar form for point, distance and angle selection - equivalent to Relative.

Related Commands

Units - sets units through the Settings dialog box.
NewWiz - sets units and other parameters for new drawings.
New - starts new drawing in Imperial or metric units.

UpdateField

Updates fields in the selected Mtext entities.

Accessing the Command

command bar: *updatefield*
menu bar: *Tools | Update Fields*
toolbar: *Redraw/Regen |*

: ***updatefield***
Prompts you in the command bar:

Select field(s) to update: select the text entities of which you want to update the fields or type all, then press Enter to select all text entities containing fields.

Press Enter to update the fields in the selected Mtext entities.

Procedures

To update a field

Related Commands

Mtext - Places paragraph text.
Field - Inserts a field in a multiline text.

Url

Opens the default Web browser (short for "uniform resource locator").

Accessing the Command

command bar: *url*
alias: *browser*

: *url*
Prompts you in the command bar:
Enter URL <http://bricsys.com>: (Enter address of a Web page.)

Command Options

Option	Description
Enter URL	Specifies a location on the Internet, such as www.bricsys.com. After you press Enter, your Web browser opens to that page. URL is short for "universal resource locator."

Related Commands

Browser - opens the default Web browser.
OnWeb - opens the Bricsys home page in your computer's default Web browser.
Mail - attaches the current drawing to a message in the default email client.

V

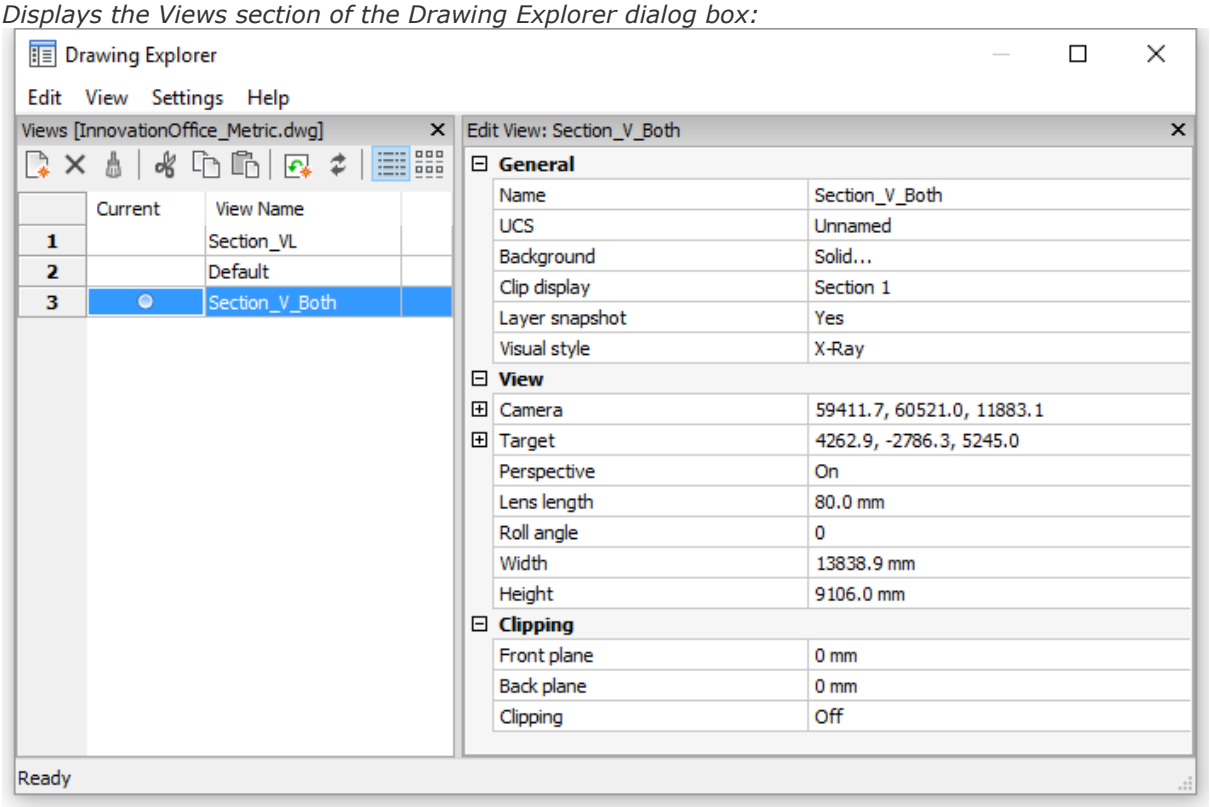
View

Creates and sets named views through the Drawing Explorer.

Accessing the Command

command bar: *view*
alias: *ddview, v, expviews*
menu bar: *Tools | Drawing Explorer | Views*

: ***view***
Displays the Views section of the Drawing Explorer dialog box:



Create or select named views, and then click X.

Press F1 to access help.

Command Options

Option	Description
Current	Designates the current view.
New	Creates new named views; clears the dialog box and then prompts you: View: ? to list saved views/Save/Window - enter an option. • ? to list saved views: Lists the names of views saved in this drawing; prompts you: View(s) to list <List all views> - enter a name or press Enter for all. BricsCAD reports like this: **Name of view: NewView1** **Space: Model**

	Height of view: 10.0830 **Center of view: X= 0.0000 Y= 0.0000** Press **F2** to see the Prompt History window. • Save: Saves the current viewpoint as the named view. Returns to the Drawing Explorer. • Window: Saves a windowed area as the view; prompts you: First corner of view window - pick a point, or enter x,y coordinates. Opposite corner - pick another point, or enter x,y coordinates. Returns to the Drawing Explorer.
Delete	Erases the named view from the drawing without warning.
Regen	Regenerates the current viewport.
Regen at each change	Toggles the regeneration of the current viewport after each change ON/OFF.
General	
View Name	Specifies the name of the view. Click to edit.
UCS	Assigns a saved UCS to the view (see the Ucs and ExpUcs commands.
Background	Sets the background for the current view or the selected saved view. Launches the Background command.
Clip Display	Allows to save the current Clip Display setting of the BIM Section planes. Optionally, click the down arrow at the right hand side of the field to select a different section plane.
Layer Snapshot	Specifies whether the current layer on/off settings are stored with the view.
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.. Click to edit.
Target	Specifies the target point of the view in x,y,z coordinates in visual perspective mode.. Click to edit.
Perspective	Sets the *Perspective* property of a view. If On, the view displays in visual perspective mode. If Off, the view displays in parallel perspective mode. A camera glyph displays in the drawing for perspective views (see the Camera command).
Width	Specifies the width of the view in current units. Click to edit.

Height	Specifies the height of the view in current units. Click to edit.
Lens Length	Defines the lens length of the camera. Click to edit.
Roll Angle	Defines the rotation of the camera around the view axis. Click to edit.
Clipping	
Front Plane	Sets the distance between the target point and the back clipping plane. Click to edit.
Back Plane	Sets the distance between the target point and the front clipping plane. Click to edit.
Clipping	Defines whether the front and/or back clipping planes are active or not. Click the down arrow button, then choose an option: • Off • Front on • Back on • Front and back on

Procedures

Exploring views
Saving a view
Restoring a view
Defining the view background
To edit the view background

Related Commands

-View - controls named views through the command bar.

Background - sets the model space background for the current viewport and saved views.
Camera - creates perspective views.
MatchPerspective - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs.
Print - plots named views.
Rename - renames views through the Drawing Explorer.
Zoom - utilizes the Previous option to show the previous view.

Explorer - Opens the Drawing Explorer dialog box.

-View

Creates and sets named views through the command bar.

Accessing the Command

command bar: *-view*
alias: *-v*
menu bar: *View | Save/Restore View*
toolbar: *View |*

: ***view***
Prompts you in the command bar:
View [? to list/Delete/Orthographic/Restore/Save/Window]: (Enter an option or type a hidden orthographic or isometric view option.)

Hidden orthographic view options: Top/Bottom/Front/Back/Left/Right

Hidden isometric view options: SWiso/SEiso/NEiso/NWiso

Command Options

Option	Description
? to list	Lists the names of views saved in this drawing; prompts you: View(s) to list <List all views> - enter a name or press Enter for all. BricsCAD reports like this: **Name of view: NewView1** **Space: Model** **Height of view: 10.0830** **Center of view: X= 0.0000 Y= 0.0000** Press F2 to see the Prompt History window.
Delete	Erases named views from the drawing; prompts you: View names to be deleted - enter one or more names. Separate multiple names with commas.
Orthographic*	Allows to switch to an orthographic view. Prompts you: **Enter an option: [Top/Bottom/Front/BAck/Left/Right]<Top>:** enter an option. If the UCSORTHO system variable is ON the related orthographic UCS is restored automatically.
Restore	Restores a named view; prompts you: View to restore - enter the name of a view.
Save	Saves the current viewpoint as the named view; prompts you: Save view as - enter a name.
Window	Saves a windowed area as the view; prompts you: First corner of view window - pick a point or enter x,y coordinates. Opposite corner - pick another point or enter x,y coordinates.

*The orientation of the orthographic and isometric views is based on the coordinate system specified by the UCSBASE system variable, which is set to the WCS (world coordinate system) by default. When one of the orthographic or isometric views is restored, BricsCAD zooms to the extents in the current viewport.

Related Commands

View - controls named views through the Drawing Explorer.
MatchPerspective - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs.
Print - plots named views.
Rename - renames views through the Drawing Explorer.
Zoom - utilizes the Previous option to show the previous view.

ViewBase

Generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

Accessing the Command

command bar: *viewbase*
menu bar: *Model | Drawing Views | Generate Drawing Views*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views | Generate Drawing Views*

: ***viewbase***
Prompts you in the command bar:

Select objects or [Entire model/preseTs] <Entire model>: (Select one or more 3D solids or press Enter to select the Entire model or select the Presets option.)

Enter new or existing layout name to make current <LayoutName>: (Press Enter to accept the current layout or type the name of a new or existing layout.)

Select position for base view [Scale/Hidden lines/Tangent lines/Orientation/Projection type/Isometric geometry/sElect]<Cancel>: (Click to position the main view or right click to abort the command.)

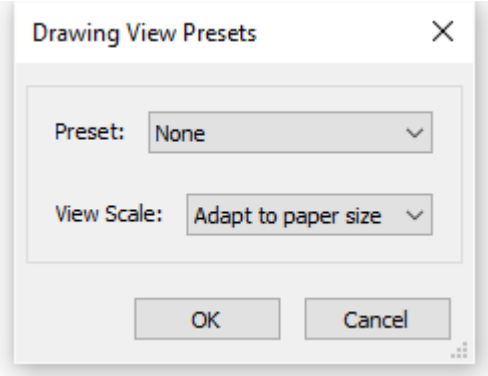
Select position for projected view <Done>: (Move the cursor left, right, above or below the base view to create an orthographic view; move the cursor diagonally with respect to the base view to create an isometric view; click to position the view or right click to stop placing views; or continue until the final view is inserted.)

NOTE The GENERATEASSOCVIEWS system variable controls whether VIEWBASE produces 2D views which are persistently associated with the source 3D model.

If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified.

Command Options

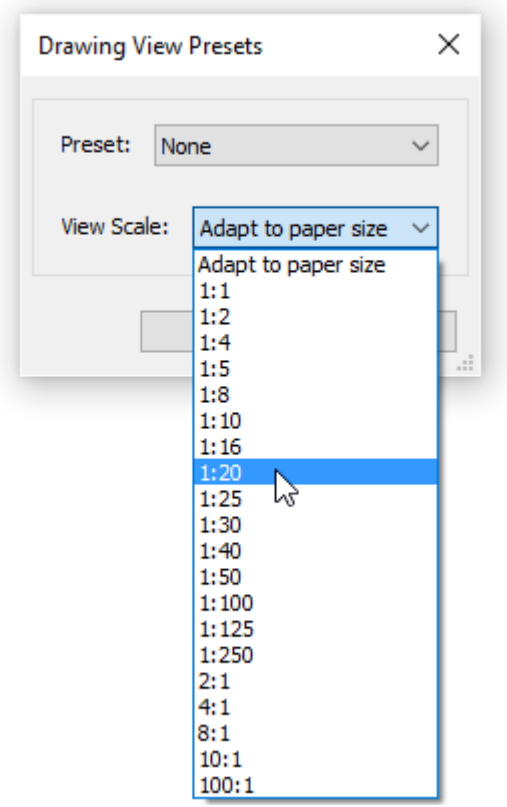
Option	Description
Select objects	Allows to compose a selection set of objects to be included in the process. Repeatedly hit the Tab-key to iterate through the entities on different nesting levels.
Presets	Allows to select a preset, which controls the number of generated drawing views and their placement in the layout. The selected preset is saved through the DRAWINGVIEWPRESET system variable. Displays a dialog box:



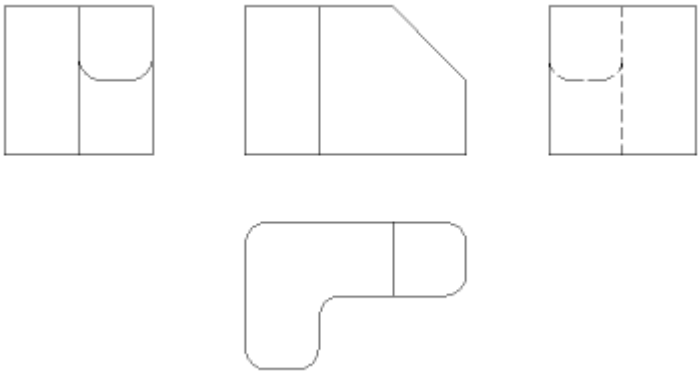
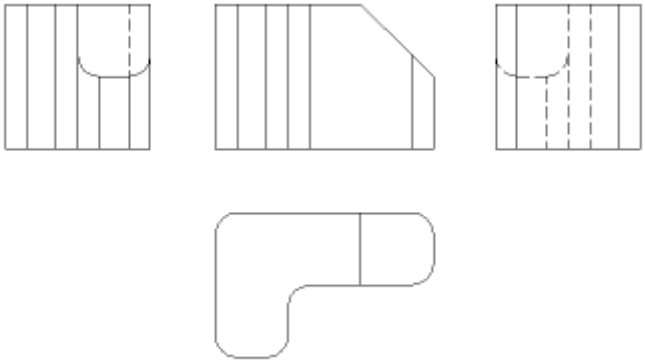
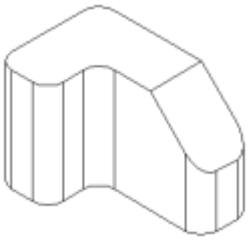
Click the *Preset* button to select a preset. The options are:

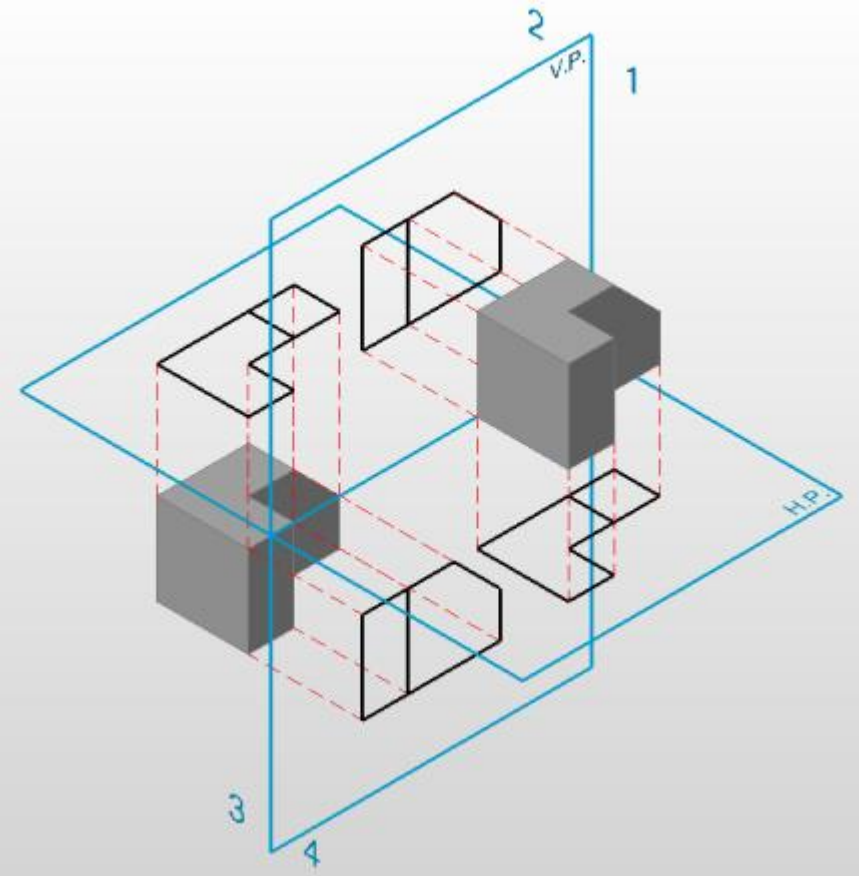
- None
- Architectural Views: generates Front, Right, Back, Left (elevations) and Top views and 3 section views: two horizontal sections (plans) and 1 vertical section.
- Mechanical Views: generates three standard views: Front, Top and Left (first angle projection) or right (third angle projection). The layout is controlled by the projection type: first or third angle. The current projection type is saved through the PROJECTIONTYPE system variable.

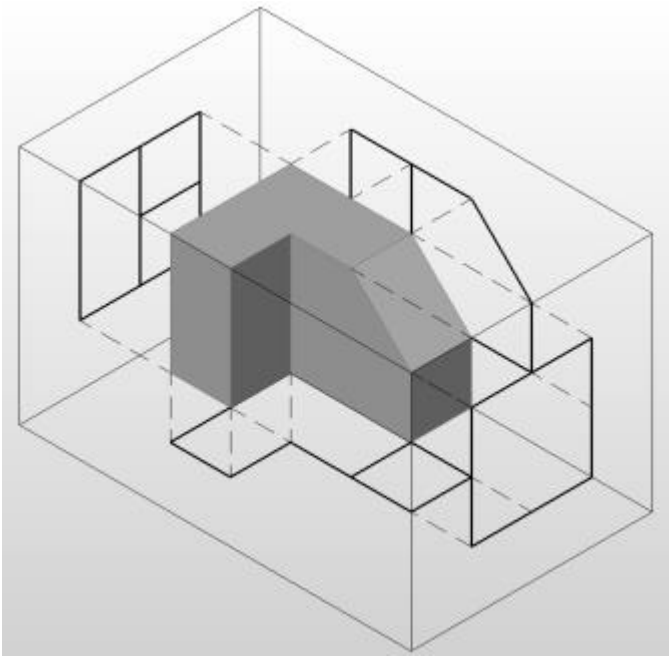
Click the View *Scale* button, then select a scale. The scale list can be edited by the SCALELISTEDIT command.

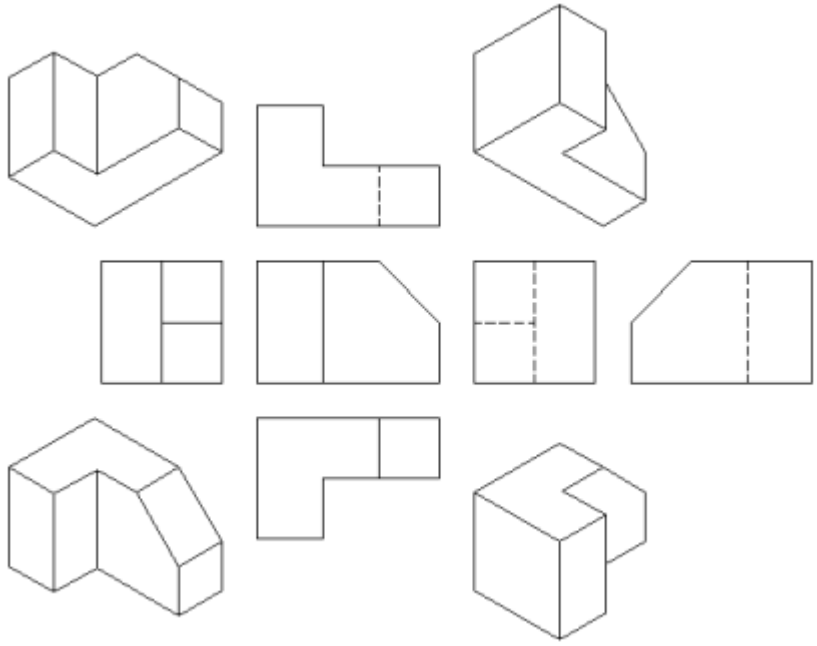


Scale	Sets the Scale property of the paper space viewports for the various views. Prompts you: Adjust view scale [fit 4 views/9 views/5 views/10 views/Standard scales/Custom] <fit 4 views>: Select an option: • fit 4 views: The scale is adjusted to fit 4 orthographic views: Front (*), Top, Left, Right. • 9 views: The scale is adjusted to fit 5 orthographic views and 4 isometric views. • 5 views: **The scale is adjusted to fit 5 orthographic views: Front (*), Top, Left, Right, Back.** • 10 views: The scale is adjusted to fit 6 orthographic views and 4 isometric views. • Standard scales: Displays the scale list as defined by the ScaleListEdit command. Select a scale in the list. • Custom: Prompts you to type a scale in the command bar. (*) The Front view (base view) is defined by the Orientation option.
Hidden Lines	Allows to control the creation of hidden lines. Hidden lines are created on the *BM_Hidden* layer. Prompts you: Enable hidden lines on basic views? [Yes/No] <Yes> - choose an option.

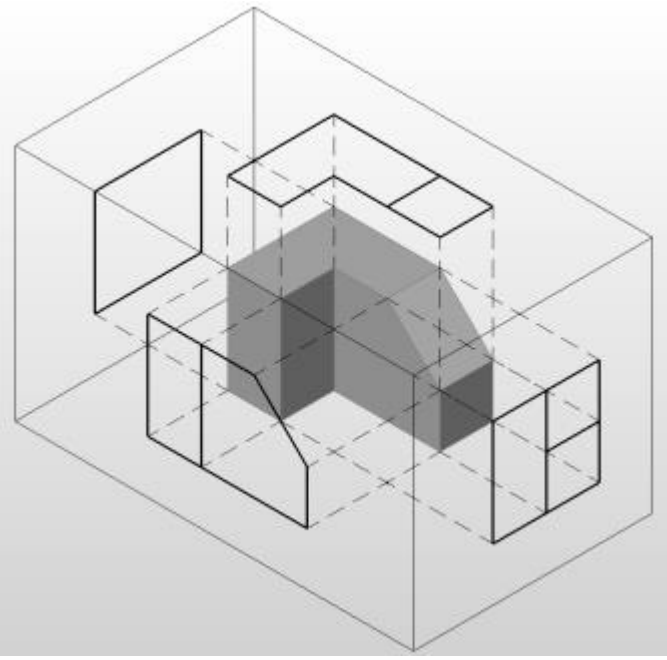
Tangent Lines	Defines whether tangent edges between tangent faces are created or not. Visible tangent edges are created on the *BM_Tangent_Visible* layer. Hidden tangent edges are created on the *BM_Tangent_Hidden* layer. Prompts you: Enable tangent edges on basic views? [Yes/No] <No>: NO:  Yes:  Tangent 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. 

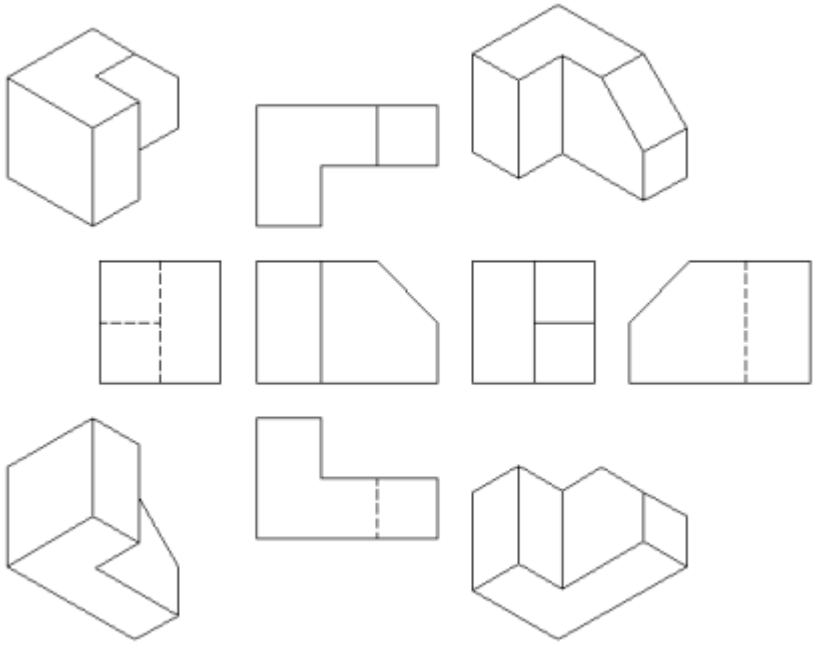
Orientation	Defines the main view. Rotates the 3D model so that the main view is projected on the vertical projection plane (V.P.).  Prompts you: Select orientation [Front/bAck/Left/Right/Top/Bottom] <Front>: Select the base view.
Projection type	Defines the layout of the views. Prompts you: Select the projection type [First angle/Third angle] <First angle>: First Angle or European projection:





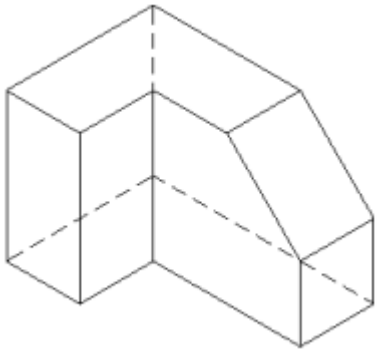
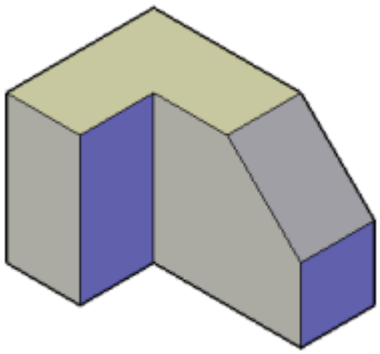
Third Angle or American projection:

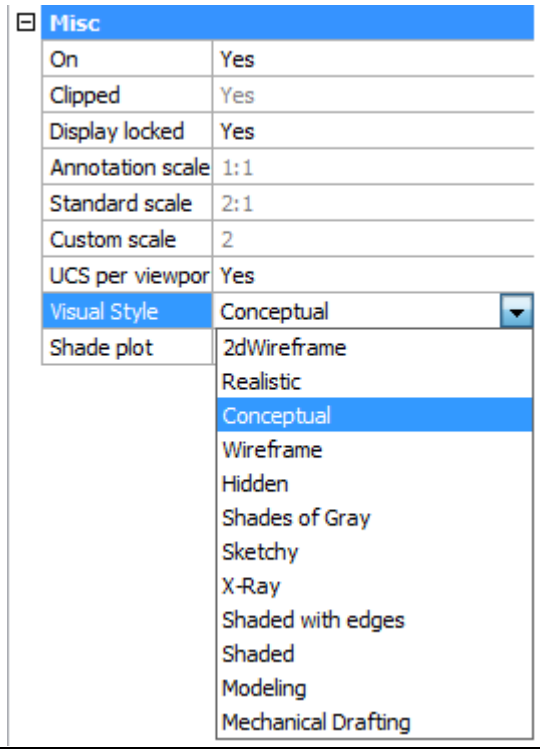




The default projection type depends on the value of the MEASUREMENT system variable:

- 0 (Imperial): Third Angle projection.
- 1 (Metric): First Angle projection.

Isometric Geometry	Defines the style for the isometric views: rendered 3D view or 2D drawing. Prompts you: Create isometric views as [2D views/3D views] <2D views>: • 2D: Isometric views are 2D drawings.  • 3D: Isometric views are 3D solids. The viewport has the *Conceptual* visual style applied.  Optionally, choose a different visual style for the viewport in the *Properties Bar*.

	
Select	Allows to select additional entities to be included or entities to be removed. Repeatedly hit the Tab-key to iterate through the entities on different nesting levels, which allows to select solids in nested components. Prompts you: Select objects to add to set or [Remove/Entire model/Layout] <Layout>: Select entities to add to drawing views or select: • Remove: select entities to be removed from the drawing views • Entire model: all entities from model space will be included in the drawing views • Layout: opens the previous layout with updated drawing views
Select position for projected view	Depending on the position of the cursor with respect to the base view one of five orthogonal views (top, left, right, back and bottom) and four isometric views can be inserted. The views are aligned automatically. Hit the Ctrl-key to toggle the alignment On/Off. If TIPS = ON in the Status Bar, the TIPS widget displays, indicating the current status of the view alignment: • Alignment ON: Views align with respect to the base view. • Alignment OFF: Views can be inserted freely.
Done	Concludes the command.

Related Commands

Flatshot - creates a hidden line representation of 3D solids in model space.
MvSetup - creates multiple paper space viewports; aligns, rotates and scales paper space viewports. In model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewProj - generates additional projected views from an existing drawing view.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

ViewDetail

Creates a detail view of a portion of a standard generated drawing at a larger scale.

Accessing the Command

command bar: *viewdetail*
menu bar: *View | Drawing Views | Detail View*
toolbars:
Drawing Views |
Detail Section Types |
ribbon: *Annotate | Drawing Views |*

: ***viewdetail***
Prompts you in the command bar:

Select drawing view: (Click inside a drawing view. Drawing view viewports highlight when you move the cursor inside their borders.)
Specify detail center or select boundary type [Circular/Rectangular] <Circular>: (Pick a point: this point is the center point of the detail view or choose an option.)

Select radius of detail view: (Pick a point or type a value to define the radius of the viewport of the detail view.)

Select position for detail view [Scale] <Cancel>: (Pick a point to position the detail view.)

Select option [Scale/Hidden lines/Tangent lines/anChor/Annotation/Boundary/modEl edge] <Cancel>: (Select an option or press Enter conclude.)

NOTE The GENERATEASSOCVIEWS system variable controls whether VIEWDETAIL produces 2D views which are persistently associated with the source 3D model.

If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified.

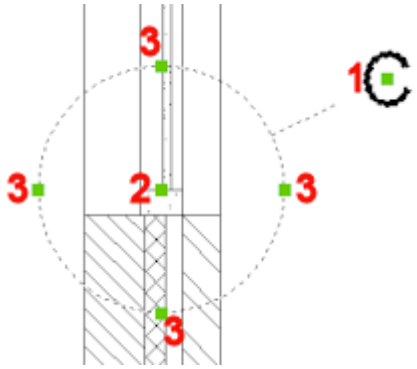
Command Options

Option	Description
Circular	Creates a circular boundary. (The Boundary option in the final prompt allows to switch the boundary to rectangular.) Prompts you: **Select radius of detail view** - Pick a point or type a value.
Rectangular	Creates a rectangular boundary. (The Boundary option in the final prompt allows to switch the boundary to circular.) Prompts you: Select radius of detail view - Pick a point or type a value.
Scale	By default the scale of the detail viewport is twice the parent viewport scale. Prompts you: **Adjust view scale [Standard scales/Custom/Relative custom/Exit] <Standard scales>:** • Standard scales: choose a standard scale in the list. The scale list can be edited by the SCALELISTEDIT command. • Custom: Allows to specify a custom scale factor. Prompts you: Enter scale <1.000000> - Enter a scale factor.

	• Relative custom: Calculates the scale factor with respect to the scale of the source view. Prompts you: Enter scale <2.000000> - Enter a number. The scale factor of the source view is multiplied by this number.
Hidden Lines	Allows to control the display of hidden lines. Prompts you: **Show hidden lines [Yes/No/from Parent]:** • Yes: Displays hidden lines. If the *BM_Ortho_Hidden* and *BM_Isometric_Hidden* layers are *OFF* or *Frozen*, hidden lines will not display. • No: Hidden lines are suppressed. • From Parent: Inherits the hidden line display from the parent view.
Tangent Lines	Allows to control the display of tangent edges between a flat and a curved face (e.g. a filleted edge). Prompts you: Show tangent lines [Yes/No]: • Yes: Tangent lines display. This is the default setting. • No: Tangent lines are suppressed.
Anchor	Allows to control how to position the detail view inside the viewport in an update operation, when the extents (= bounding box) of the 3D geometry has been changed. Prompts you: Anchor view on paper space? [Yes/No]: • Yes: The center of the viewport is anchored. The size of the viewport grows or shrinks around its center point. • No: The position of the unchanged geometry is fixed. This is the default behavior. The AUTOVPFITTING system variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
Annotation	Controls the annotation of the detail view . Prompts you: **Select annotation [Identifier/Label]:** • Identifier: Prompts you to specify the view detail identifier: Specify the identifier <A>: Enter a name for the view detail. • Label: Prompts you to display of the view detail label. Show view label [Yes/No]:
Boundary	Controls the boundary if the detail view : circular or rectangular. Prompts you: Detail boundary [Rectangular/Circular]: Choose an option.
Model Edge	Controls whether a connection line is drawn between the detail view and the detail boundary in the parent view. Prompts you: Model edge type [smooth with Border/smooth with Connection line]: Choose an option.

Grips Editing

Detail views can be edited through grips.
6 grips display:



1. Controls the position of the identifier
2. Controls the position of the section detail.
3. 4 grips allow to control the size of the detail boundary.

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewProj - generates additional projected views from an existing drawing view.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

ViewDetailStyle

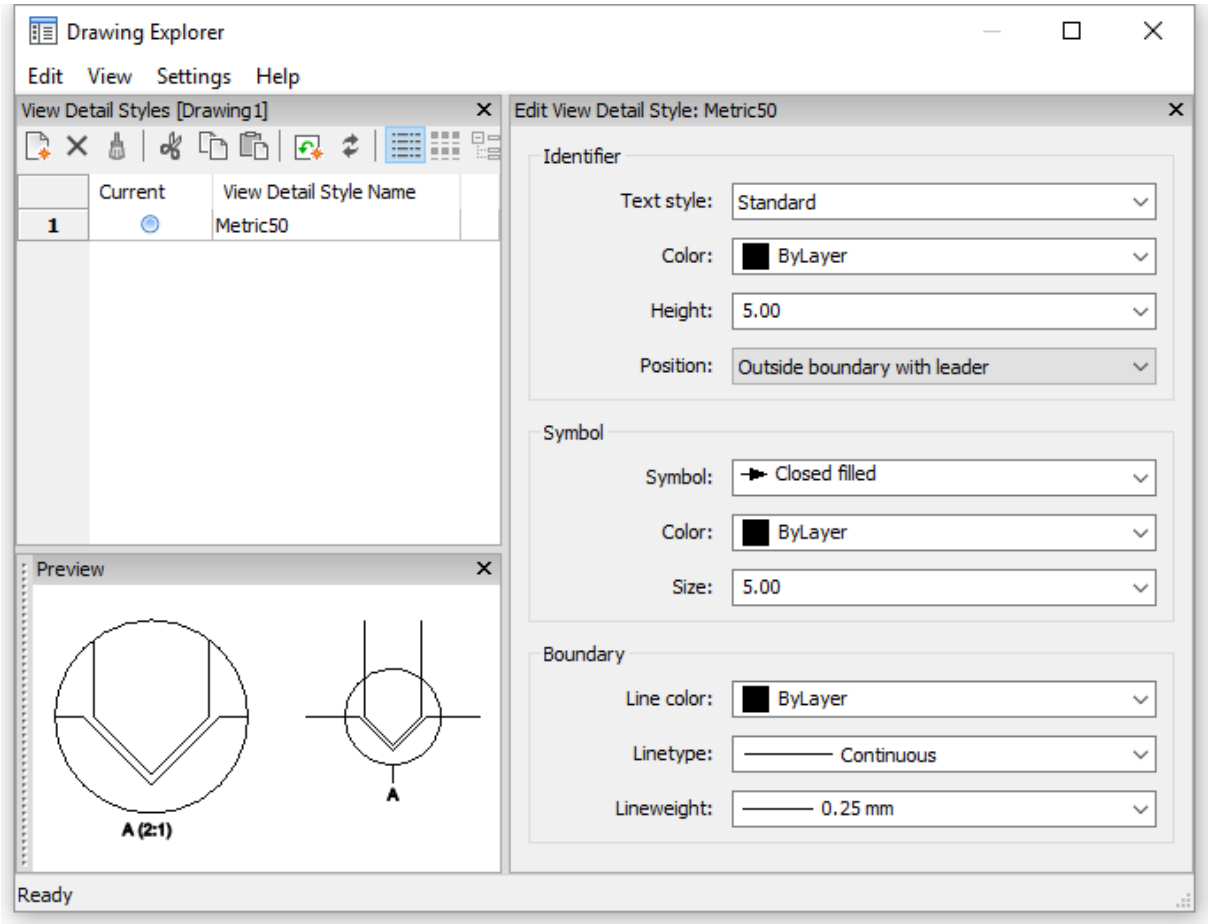
Allows to specify the visual format of detail views and detail symbols.

Accessing the Command

command bar: *viewdetailstyle*
menu bar: *View | Drawing Views | Detail Style*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views|*

: ***viewdetailstyle***

Displays the View Detail Styles section of the Drawing Explorer dialog box:



Command Options

Option	Description
New	Creates a new view detail style as a copy of the currently selected style.

Identifier	Defines the properties of the view detail identifier. Identifier Text style: Standard Color: ByLayer Height: 5.00 Position: Outside boundary with leader
Symbol	Defines the properties of the symbol. Symbol Symbol: Closed filled Color: ByLayer Size: 5.00
Boundary	Defines the properties of the detail viewport boundary. Boundary Line color: ByLayer Linetype: Continuous Lineweight: 0.25 mm

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewSection - creates a cross section view based on a standard drawing view generated by the ViewBase command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.

ViewEdit

Allows changing the scale and the hidden line visibility of drawing views.
The command can be used in paper space only.

Accessing the Command

command bar: *viewedit*
menu bar: *Model | Drawing Views | Edit View*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views| Edit View*

: ***viewedit***
Prompts you in the command bar:

Select drawing views/<All views>: (Click inside one or more drafting viewports, then press Enter.)
The next prompt depends on the selected view type. If different view types are selected, only the shared options are available.

Standard views:

Select option [Scale/Hidden lines/Tangent lines/anChor/Geometry/sElect*] <Cancel>: (Select an option)

Section views:

Select option [Scale/Hidden lines/Tangent lines/anChor/Geometry/Annotation/Depth/Projection] <Cancel>: (Select an option)

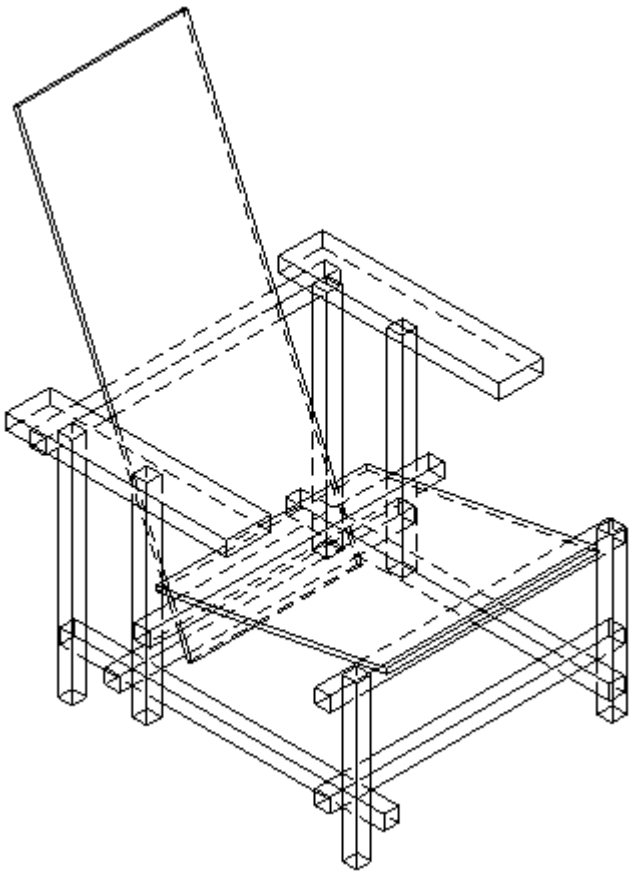
Detail views:

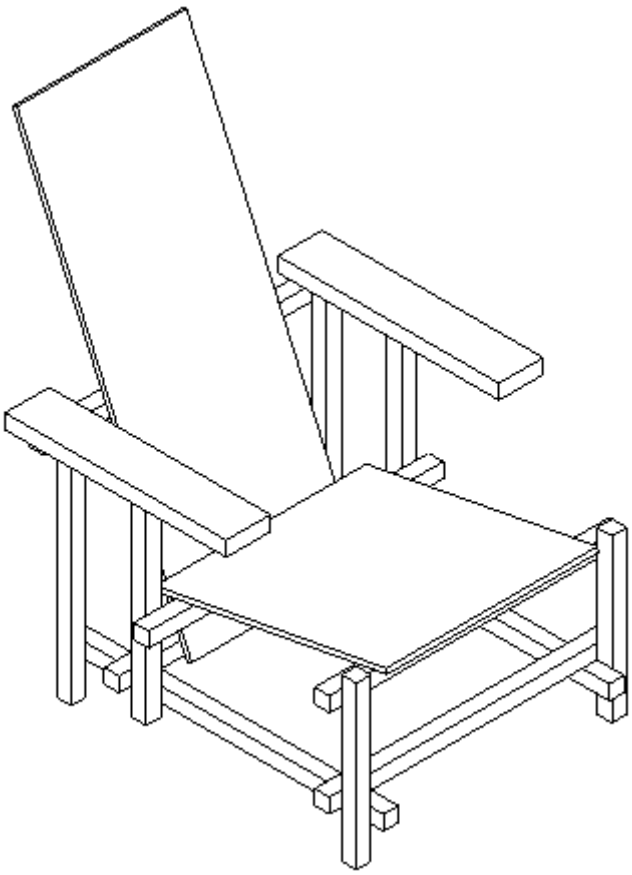
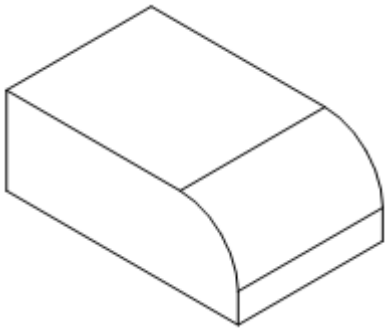
Select option [Scale/Hidden lines/Tangent lines/anChor/Annotation/Boundary/modEl edge] <Cancel>: (Select an option)+

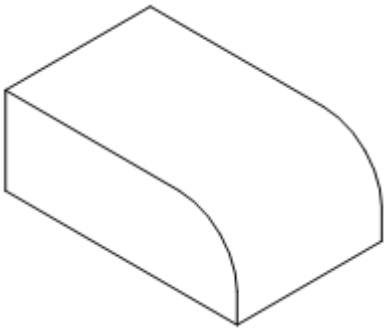
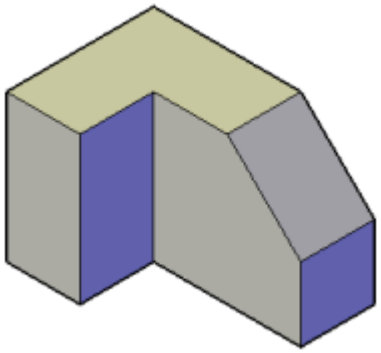
* This option is available when a single view is selected only.

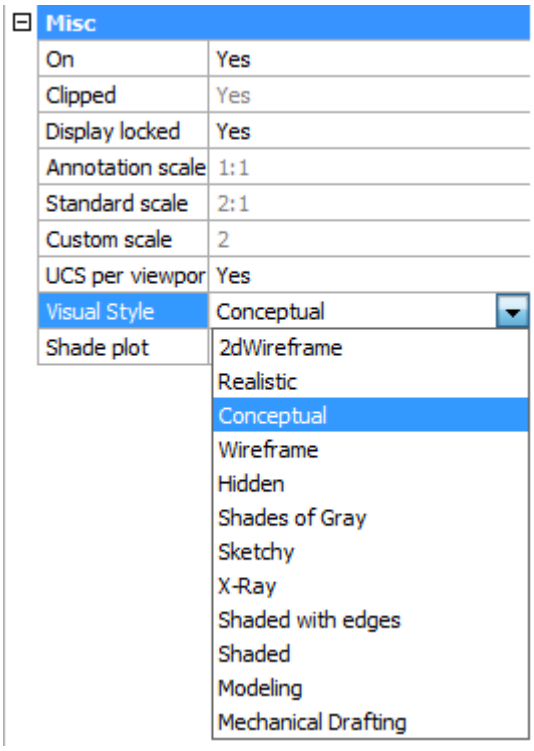
Command Options

Option	Description
Scale	Allows to modify the Scale (Standard Scale or Custum Scale) property of a drafting viewport. Prompts you in the command bar: Adjust view scale [Standard scales/Custom]: • Standard scales: Type a scale number, then press Enter or choose a standard scale in the prompt menu. • Custom: Type a value in the command bar, then press Enter.
Hidden Lines	Allows to control the visibility of hidden lines. Prompts you in the command bar: Show hidden lines [Yes/No]: • Yes: Displays hidden lines in the selected drafting viewports. If the *BM_Ortho_Hidden* and *BM_Isometric_Hidden* layers are *OFF* or *Frozen*, hidden lines will not display.

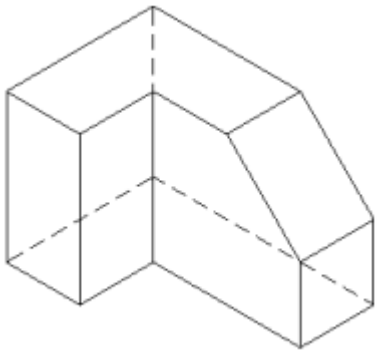
	 • No: Switches the display of hidden lines in the selected drafting viewports off.

	
Tangent Lines	Allows to control the display of tangent edges between a flat and a curved face (e.g. a filleted edge). Prompts you: Show tangent lines [Yes/No]: • Yes: Tangent lines display. This is the default setting. 

	• No: Tangent lines are suppressed. 
Anchor	Allows to control how to position the view inside the viewport in an update operation, when the extents (= bounding box) of the 3D geometry has been changed. Prompts you: Anchor view on paper space? [Yes/No]: • Yes: The center of the viewport is anchored. The size of the viewport grows or shrinks around its center point. • No: The position of the unchanged geometry is fixed. This is the default behavior. The AUTOVPFITTING system variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
Geometry	Defines how the geometry displays: 3D or 2D. Prompts you: Choose geometry type [2D/3D]:2D • 3D: Geometry is 3D. Applies the *Conceptual* visual style.  Optionally, choose a different visual style for the viewport in the *Properties Bar*.

<table>
<tr>
<td></td>
<td>


2D: Geometry is 2D.


</td>
</tr>
<tr>
<td>Select</td>
<td>
Allows to select additional entities to be added or removed from the view group. This option is available when a single view is selected only. The ViewBase command generates a group of associated views; changes applied to a view of a such group will also be applied to the other members.

Repeatedly hit the Tab-key to iterate through the entities on different nesting levels, which allows to select solids in nested components.
Prompts you:
Select objects to add to set or [Remove/Entire model/Layout] <Layout>:
Select entities to add to drawing views or select:

Remove: select entities to be removed from the drawing views
Entire model: all entities from model space will be included in the drawing views
Layout: opens the previous layout with updated drawing views

</td>
</tr>
</table>

Annotation	Controls the annotation of the detail view or section view. Prompts you: **Select annotation [Identifier/Label]:** • Identifier: Prompts you to specify the view detail identifier: Specify the identifier <A>: Enter a name for the view detail. • Label: Prompts you to display of the view detail label. Show view label [Yes/No]: Choose an option.
Depth	Specifies the view depth of a section view. Prompts you: Choose depth mode [Full/Custom]: • Full: The view depth is set to the extents of the model (= maximum view depth). • Custom: Allows to limit the view depth. Prompts you: Enter depth distance: Type a positive value then press Enter. or Define the view depth dynamically using the pointing device: a line, indicating the view depth, displays dynamically. Click a point.
Projection	The command prompts you: Select projection method [Normal/Orthogonal]: Select the desired method: • Normal: Creates the section as a union of projections of every cutting region (each segment of a the section polyline) in its normal direction. • Orthogonal: Creates the section in normal direction of the first section region (first segment of the section polyline).
Boundary	Controls the boundary if a detail view : circular or rectangular. Prompts you: Detail boundary [Rectangular/Circular]: Choose an option.
Model Edge	Controls whether a connection line is drawn between the detail view and the detail boundary in the parent view. Prompts you: Model edge type [smooth with Border/smooth with Connection line]: Choose an option.

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewProj - generates additional projected views from an existing drawing view.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

ViewExport

Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
This command can be used in paper space only.

Accessing the Command

command bar: *viewexport*
menu bar: *View| Drawing Views | Export Drawing Views to Model Space*
toolbar: *View | Drawing Views |*
ribbon: *Annotate | Drawing Views|*

: ***viewexport***
Prompts you in the command bar:

Select drawing views/<All views>: (Select views or press Enter to export all views.).

Select drawing views/<All views>: (Select more views then press Enter to stop selecting views.)
Choose destination [Clipboard/File/Model space] <Model space>: (Select an option.)

Command Options

Option	Description
Select drawing views	Click inside a paper space viewport. Selected viewports display in dashed lines.
All views	All views in the current layout will be exported.
Choose destination	Allows to specify the destination for the view export
Clipboard	Exports the selected view(s) to the clipboard. Allows to paste the view(s) in another drawing.
File	Displays a *File* dialog. Enter a name in the *File Name* field, then press the *Save* button.
Model space	Remove exported views? <No>/Yes: (Select an option.) • *Yes*: Viewports of exported views are removed from the layout. • No: Viewports of exported views are kept in the layout. Model space displays. Select position for exported views: **(Specify a point.)**

Related Commands

Flatshot - creates a hidden line representation of 3D solids in model space.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewProj - generates additional projected views from an existing drawing view.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

ViewProj

Generates additional projected views from an existing drawing view.

Accessing the Command

command bar: *viewproj*
menu bar: *View| Drawing Views | Projected Views*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views|*

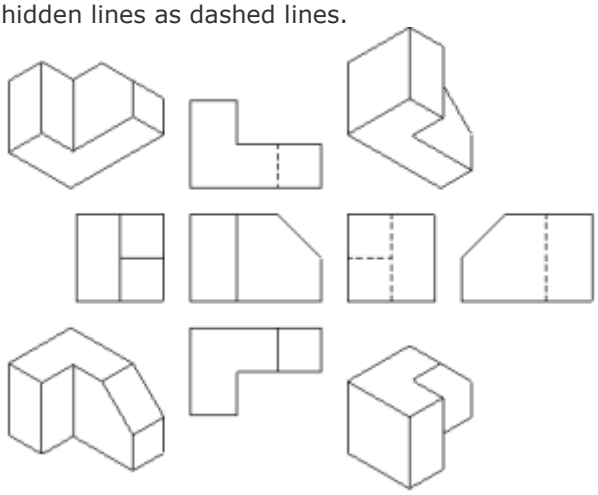
: ***viewproj***
Prompts you in the command bar:

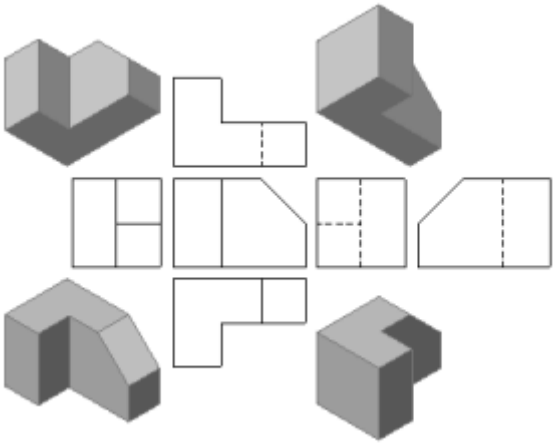
Select drawing view: (Click inside an existing drawing view.)

Select position for projected view [Isometric geometry] <Cancel>: (Position the projected views with respect to the existing drawing view. Right click to stop adding projected views.)

4 orthogonal and 4 isometric views can be generated from an existing drawing view.

Command Options

Option	Description
Select position of projected view	Depending on the position of the cursor with respect to the selected view one of five orthogonal views and four isometric views can be inserted. The views are aligned automatically. Hit the Ctrl-key to toggle the alignment On/Off. If TIPS = ON in the Status Bar, the TIPS widget displays, indicating the current status of the view alignment.
Isometric geometry	Sets the style for isometric views. Prompts you: **Create isometric views as [3D views / 2D views] <2D>:** The options are: • 2D views: Isometric views are created as a 2D projection of the 3D geometry. Set the display of the *BM_Isometric_Hidden* layer on to display hidden lines as dashed lines.  • 3D views: Isometric views show the 3D geometry.

	• The *Visual Style* property of the viewport controls how isometric views display. 

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

ViewRes

Sets the view resolution and toggles fastzoom mode (short for "view resolution").

Accessing the Command

command bar: *viewres*

: ***viewres***
Prompts you in the command bar:

Do you want fast zooms? <Yes>: (Type Y or N.)

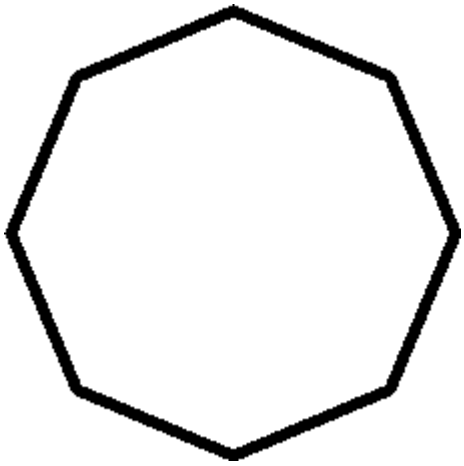
Enter curve display quality (range 1-20000) <100>: (Enter a number, or press Enter to accept the default.)

This command was useful when computers and graphics boards were slow.

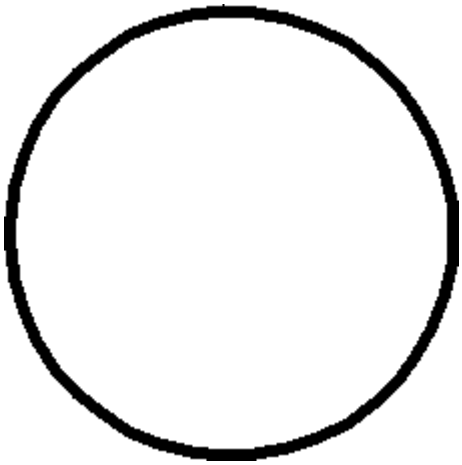
It allowed AutoCAD drawings to be displayed more quickly at the cost of lower display resolution.

This command is no longer necessary.

When ViewRes is set to low values, zoomed circles can look like polygons (low display resolution):



When ViewRes is set to high values, or when the Regen command is used, the circles look round again (high display resolution):



Command Options

Option	Description
Do you want fast zooms?	Specifies how redraws are handled: • Yes - redraws and other commands that activate redraws refresh the drawing screen. • No - redraws acts like regens: they read the drawing database and then refresh the screen. This option is included only for compatibility with AutoCAD prior to Release 2.5.
Enter curve display quality	Specifies the roundness of circles and other curved entities. Enter a number between 1 and 20000: • 1 - very fast display, but circles can look like polygons. • 100 - default value. • 20000 - slower display, but circles almost always look round. With today's fast computers and graphics boards, you can safely enter 20000.

Related Command

Regen - regenerates the drawing.

ViewSection

Creates a cross section view based on a standard drawing view generated by the ViewBase command in a paper space layout.

Accessing the Command

command bar: *viewsection*
menu bar: *Model | Drawing Views | Generate Section*
toolbars:
Drawing Views |
Section types |
ribbon:
Annotate | Drawing Views|

: ***bmgensection***
Prompts you in the command bar:

Select drawing view: (Select a drawing view in the current paperspace layout.)

Specify start point of section line or select type [Full/Half/Offset/Aligned] <Full>: (Pick a point or select an option.)

Specify next point of section line [Done] <Done>: (Pick a point.)

The section view displays dynamically parallel to the section line. The view direction of the section view is defined by the position of the section view with respect to the section line.
Select position for section view: *(Pick a point.)*

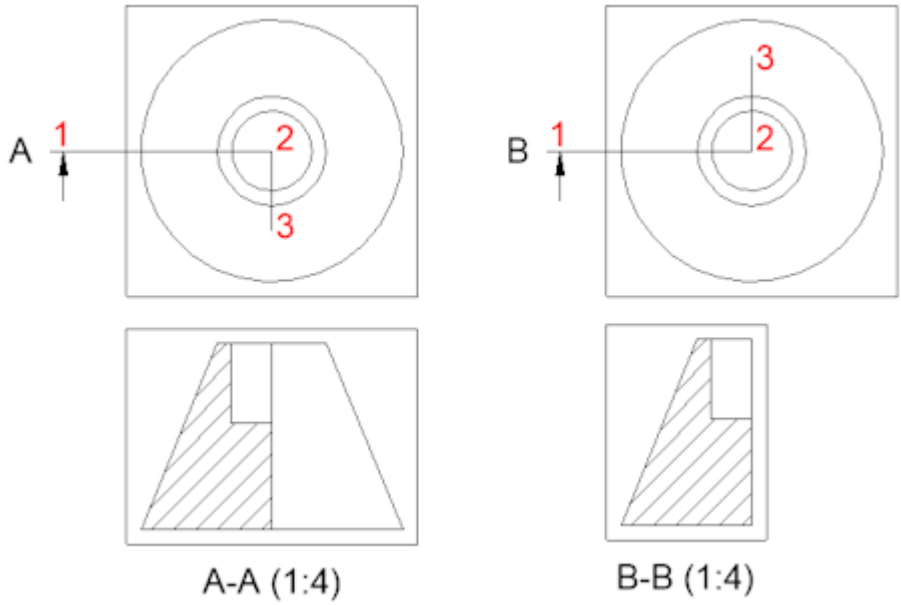
Select option [Scale/Hidden lines/Tangent lines/anChor/Geometry/Annotation/Depth/Projection] <Cancel>: (Press <Enter> to finish or select an option.)

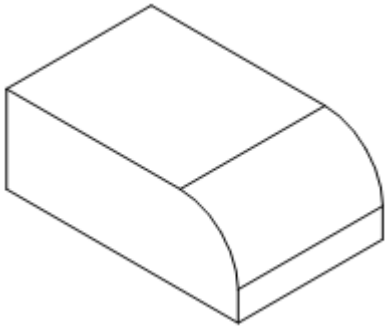
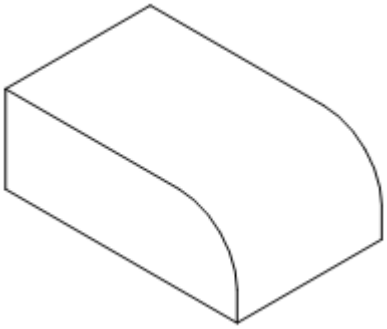
NOTES

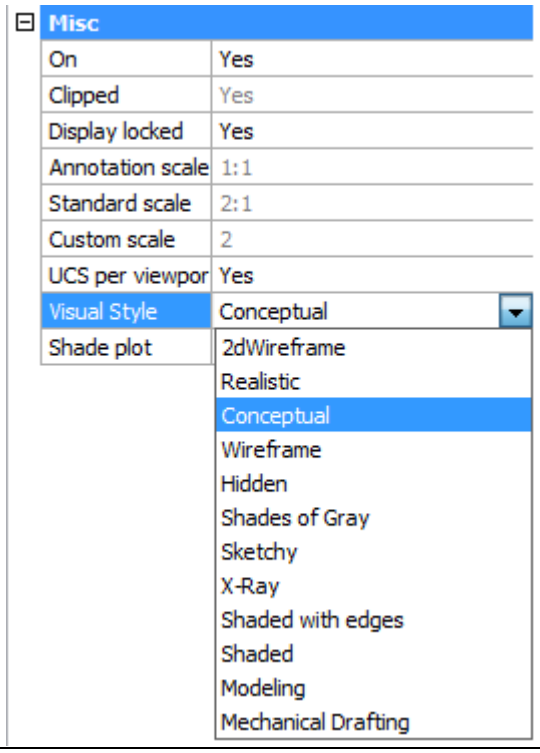
- The GENERATEASSOCVIEWS system variable controls whether VIEWSECTION produces 2D views which are persistently associated with the source 3D model. If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified.
- If the Sectionable property of a mechanical component is OFF, the component appears non-sectioned in the section views of the Full section type.

Command Options

Option	Description
Type	Allows to control the shape of the section plane. Prompts you: Select type [Full/Half/Offset/Aligned] <Full>: Select an option. • Full ( ): The section line defines an infinite plane cutting through the entire model. • Half ( ): The section line defines a half-plane cutting part of the model. Prompts you: Specify start point of section line or [Type] <Type>: Specify a point (1). Specify next point of section line [Done] <Done>: Specify a point to define the limit of the half plane (2). Specify next point of section line [Done] <Done>: Specify a point to define the view direction (3). The position of the third point controls whether the part of the model that is not cut displays (left) or not (right).

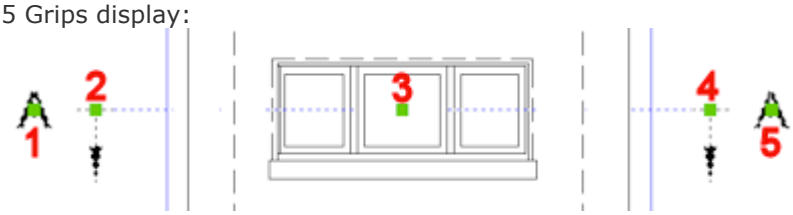
	 • Offset ( ): The section line defines a series of cutting regions located on different offsets from each other. You have to chose *Done* from the option menu to complete the section line. • Aligned ( ): the section line defines a polyline, where every segment defines a cutting region. The resulted section will have the length equal to the sum of the lengths of cutting regions. You have to chose *Done* in the option menu to complete the section line.
Select position for section view	When placing the section view moves perpendicular to the section line. The view direction is defined by the position of the cursor with respect to the section line. Hit the Ctrl-key to toggle the alignment On/Off. If TIPS = ON in the Status Bar, the TIPS widget displays, indicating the current status of the view alignment.
Scale	Sets the Scale property of the paper space viewports for the various views. Prompts you: Adjust view scale [Standard scales/Custom/from Parent]: Select an option: • Standard scales: Displays the scale list as defined by the ScaleListEdit command. Select a scale in the list. • Custom: Prompts you to type a scale in the command bar. • From Parent: Sets the scale of the section equal to the scale of the parent viewport.
Hidden Lines	Allows to control the visibility of hidden lines. Prompts you in the command bar: Show hidden lines [Yes/No/from Parent]: • Yes: Displays hidden lines in the selected drafting viewports. If the *BM_Ortho_Hidden* and *BM_Isometric_Hidden* layers are *OFF* or *Frozen*, hidden lines will not display. • No: Switches the display of hidden lines in the selected drafting viewports off. • From Parent: Copies the Hidden Lines property from the parent viewport.

Tangent Lines	Allows to control the display of tangent edges between a flat and a curved face (e.g. a filleted edge). Prompts you: Show tangent lines [Yes/No]: • Yes: Tangent lines display. This is the default setting.  • No: Tangent lines are suppressed. 
Anchor	Allows to control how to position the view inside the viewport in an update operation, when the extents (= bounding box) of the 3D geometry has been changed. Prompts you: Anchor view on paper space? [Yes/No]: • Yes: The center of the viewport is anchored. The size of the viewport grows or shrinks around its center point. • No: The position of the unchanged geometry is fixed. This is the default behavior. The AUTOVPFITTING system variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
Geometry	Defines the visual style for the section view: rendered 3D view or 2D drawing. Prompts you: Choose geometry type [2D/3D]: • 2D: The section view uses the 2dWireframe visual style. • 3D: The section view uses a rendered visual style, which is *Conceptual* by default. Optionally, choose a different visual style for the viewport in the *Properties Bar*.

	
Annotation	Controls the annotation of the detail view or section view. Prompts you: **Select annotation [Identifier/Label]:** • Identifier: Prompts you to specify the view detail identifier: Specify the identifier <A>: Enter a name for the view detail. • Label: Prompts you to display of the view detail label. Show view label [Yes/No]: Choose an option.
Depth	Specifies the view depth of a section view. Prompts you: Choose depth mode [Full/Custom]: • Full: The view depth is set to the extents of the model (= maximum view depth). • Custom: Allows to limit the view depth. Prompts you: Enter depth distance: Type a positive value then press Enter. or Define the view depth dynamically using the pointing device: a line, indicating the view depth, displays dynamically. Click a point.
Projection	The command prompts you: Select projection method [Normal/Orthogonal]: Select the desired method: • Normal: Creates the section as a union of projections of every cutting region (each segment of a the section polyline) in its normal direction. • Orthogonal: Creates the section in normal direction of the first section region (first segment of the section polyline).

Grips Editing

Section lines can be edited through grips.
Select 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 to move the section line.
4. Defines the endpoint of the section line.
5. Controls the position of the second identifier.

Related Commands

Flatshot - creates a hidden line representation of 3D solids in model space.
Solprof - creates a hidden line representation of 3D solids in a layout viewport.
MvSetup - Creates multiple paper space viewports; aligns, rotates and scales paper space viewports. In model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.
ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewProj - generates additional projected views from an existing drawing view.
ViewSectionStyle - allows to specify the visual format of section views and section lines.
ViewUpdate - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

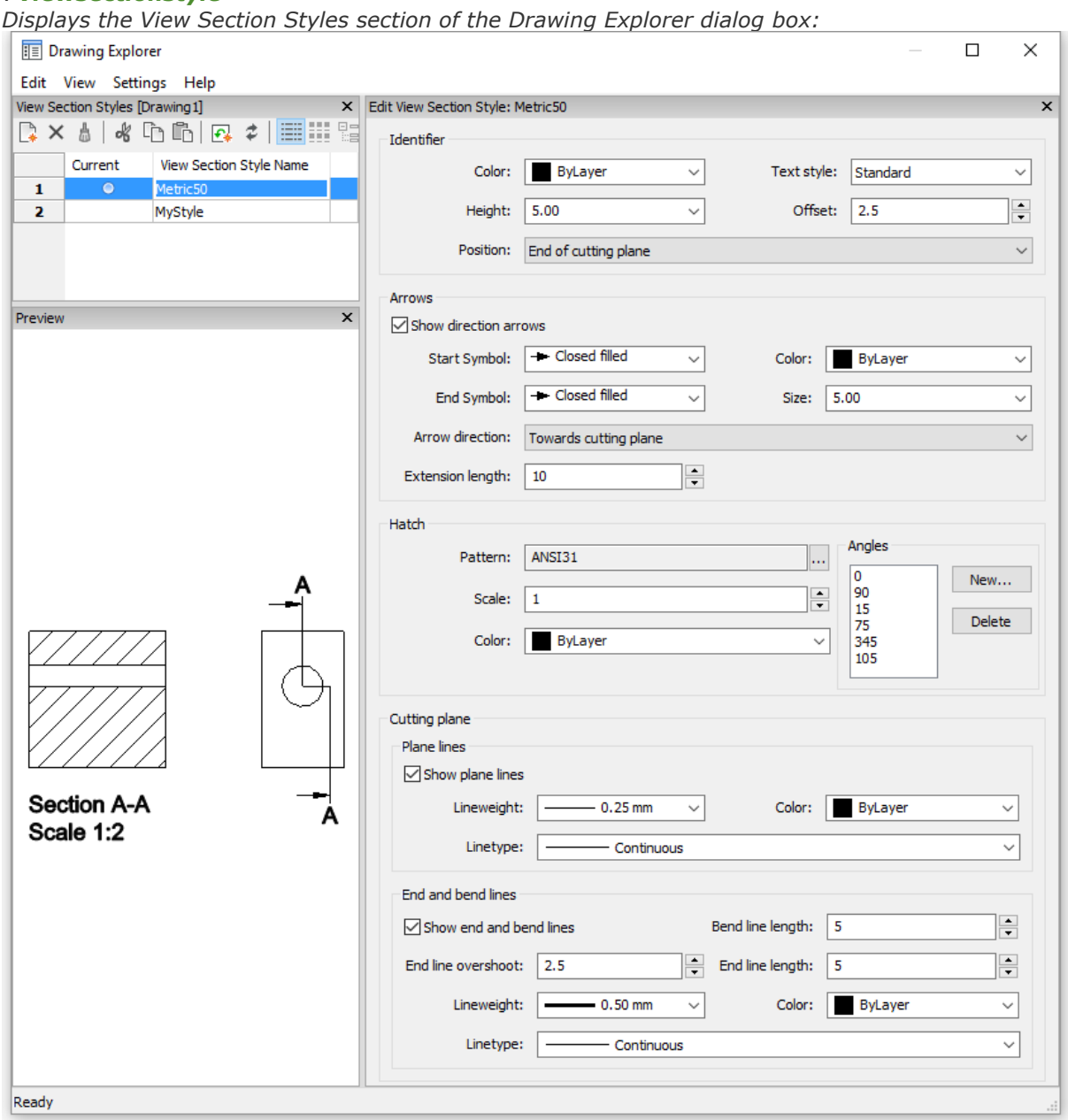
ViewSectionStyle

Allows to specify the visual format of section views and section lines.

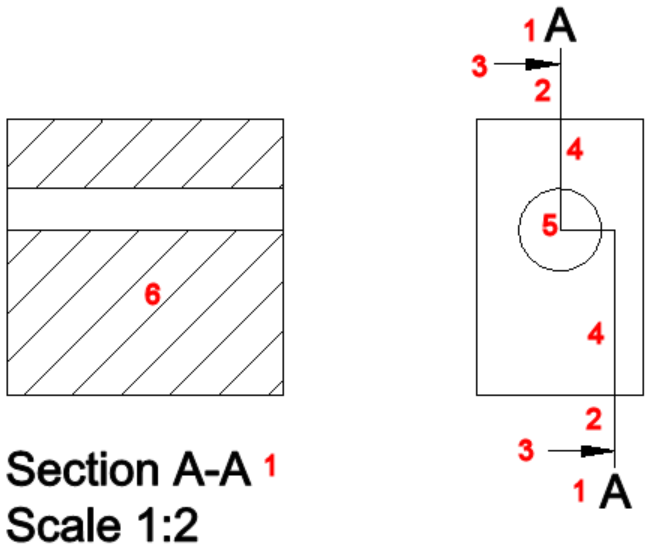
Accessing the Command

command bar: *viewsectionstyle*
menu bar: *View| Drawing Views | Section Style*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views|*

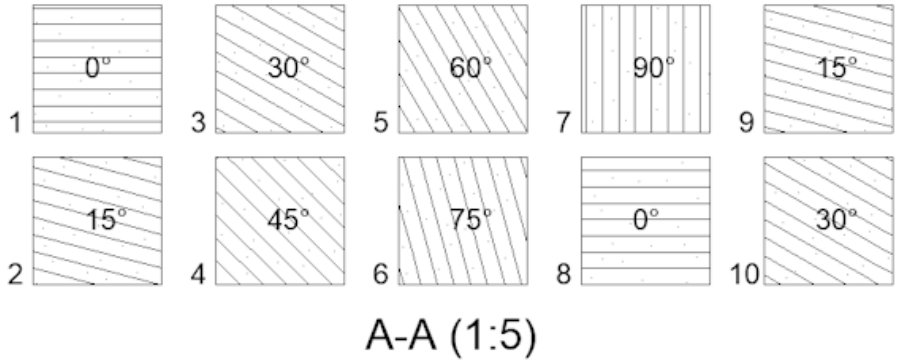
: ***viewsectionstyle***
Displays the View Section Styles section of the Drawing Explorer dialog box:



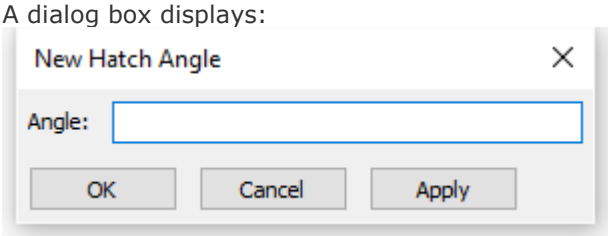
Command Options

Option	Description
New	Creates a new view section style as a copy of the currently selected style.
	 1. Identifier 2. Arrows 3. Allows extension length 4. Cutting plane lines 5. Cutting plane bend lines 6. Hatch
Identifier	Sets the properties of the section identifier: Identifier Color: ByLayer Text style: Standard Height: 5.00 Offset: 2.5 Position: End of cutting plane
Arrows	Sets the properties of the section arrows: Arrows Show direction arrows Start Symbol: Closed filled Color: ByLayer End Symbol: Closed filled Size: 5.00 Arrow direction: Towards cutting plane Extension length: 10
Hatch	Sets the properties of the section hatching:

The first angle in the list is applied to the first solid in the model, the second angle to the second solid, the third angle to the third solid, etc. In the dialog above seven angles are defined, in a model containing 10 solids the hatch angles are applied as indicated in the section below. Because only 7 angles are defined, the first angle is applied to solid no. 8, the second angle to no. 9 etc.

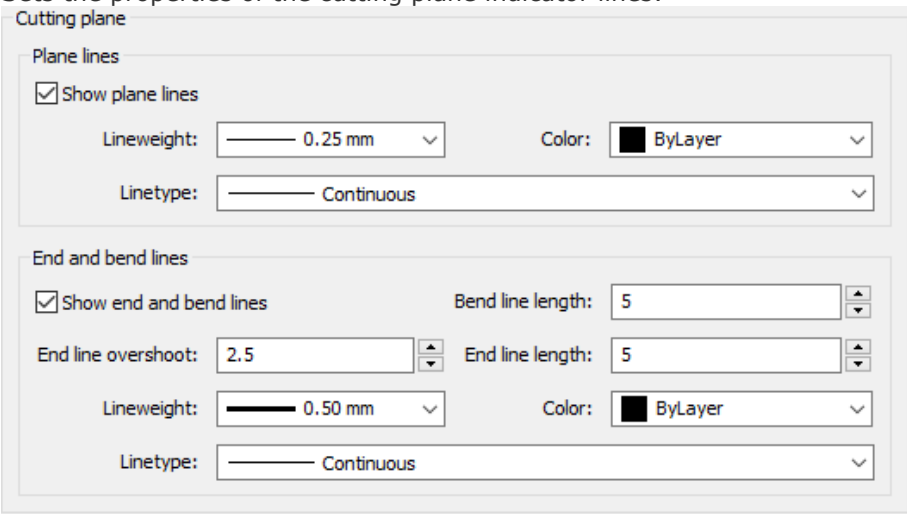


To add a new hatch angle, click the *New...* button.
A dialog box displays:



- Type a value in the *Angle* field.
- Do one of the following:
  - Click the *OK* button to add the angle and close the dialog.
  - Click the *Apply* button to add the angle and keep the dialog open to add more angles.
  - Click the *Cancel* button to close the dialog without adding an angle.

To remove an angle, select the angle in the list, then click the *Delete* button.

Cutting Plane	Sets the properties of the cutting plane indicator lines: 

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewSection - creates a cross section view based on a standard drawing view generated by the ViewBase command in a paper space layout.

ViewUpdate

Updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION when VIEWUPDATEAUTO = 0.

Accessing the Command

command bar: *viewupdate*
menu bar: *Model | Drawing Views | Update Drawing Views*
toolbar: *Drawing Views |*
ribbon: *Annotate | Drawing Views|*

: ***viewupdate***
Prompts you in the command bar:

Select drawing views/<All views>: (Click inside a view you want to be updated or press Enter to update all views.)

Select drawing views/<All views>: (Click inside a view you want to be updated or press Enter to update the selected view(s).)

NOTE Click the *ViewUpdateAuto* tool button ( ) on the *Drawing Views* toolbar or the *Annotate | Drawing Views* ribbon panel to toggle the VIEWUPDATEAUTO system variable ON/OFF.
The state of the tool button indicates the current value of the VIEWUPDATEAUTO system variable: ON ( ) or OFF ( ).

Command Options

Option	Description
All views	Updates all drawing views in the current layout. Do one of the following: • Press Enter at the first prompt. • Type A at the second and following prompts.

Related Commands

ViewBase - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
ViewExport - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.
ViewDetail - creates a detail view of a portion of a standard generated drawing at a larger scale.
ViewDetailStyle - allows to specify the visual format of detail views and detail symbols.
ViewEdit - allows changing the scale and the hidden line visibility of drawing views.
ViewExport - Exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing or to a new drawing.
ViewProj - generates additional projected views from an existing drawing view.
ViewSection - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
ViewSectionStyle - allows to specify the visual format of section views and section lines.

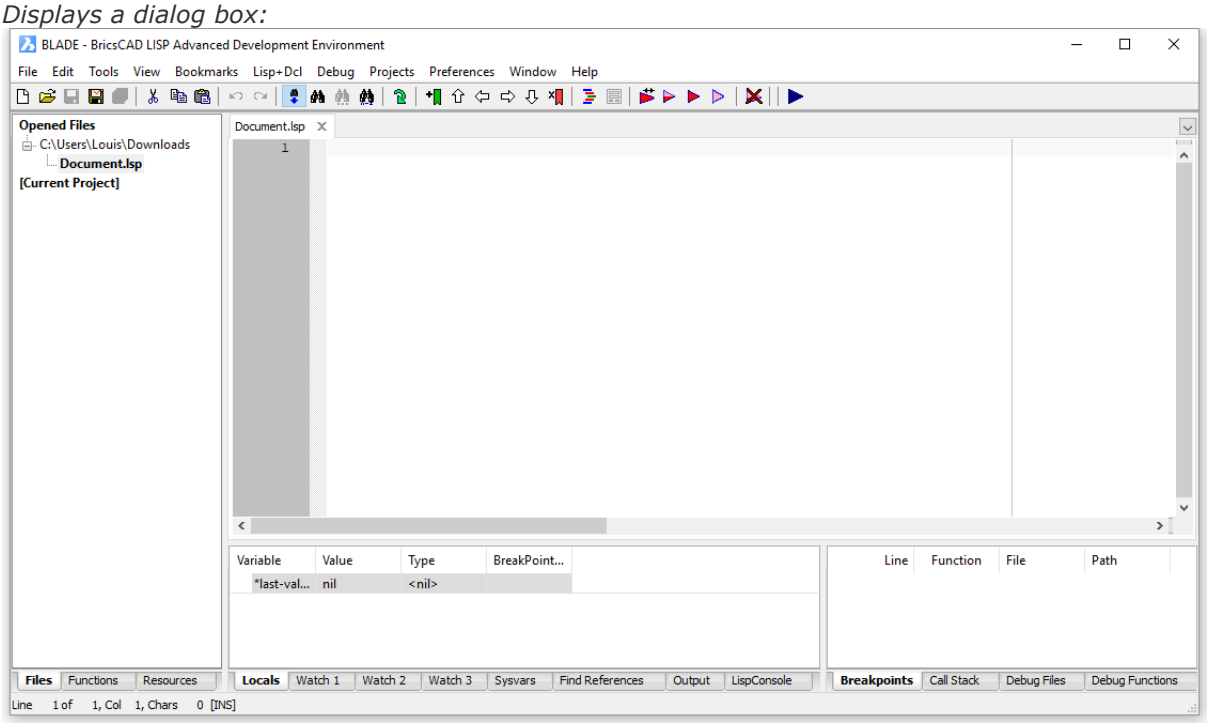
VLIDE

Opens the LISP Development Environment for editing and debugging LISP applications.

Accessing the Command

command bar: *vlide*

: ***vlide***
Displays a dialog box:



Command Options

No command options

Related Commands

BLADE - Opens the advanced environment for editing and debugging LISP applications.

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
VbaLoad loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

VisualStyles

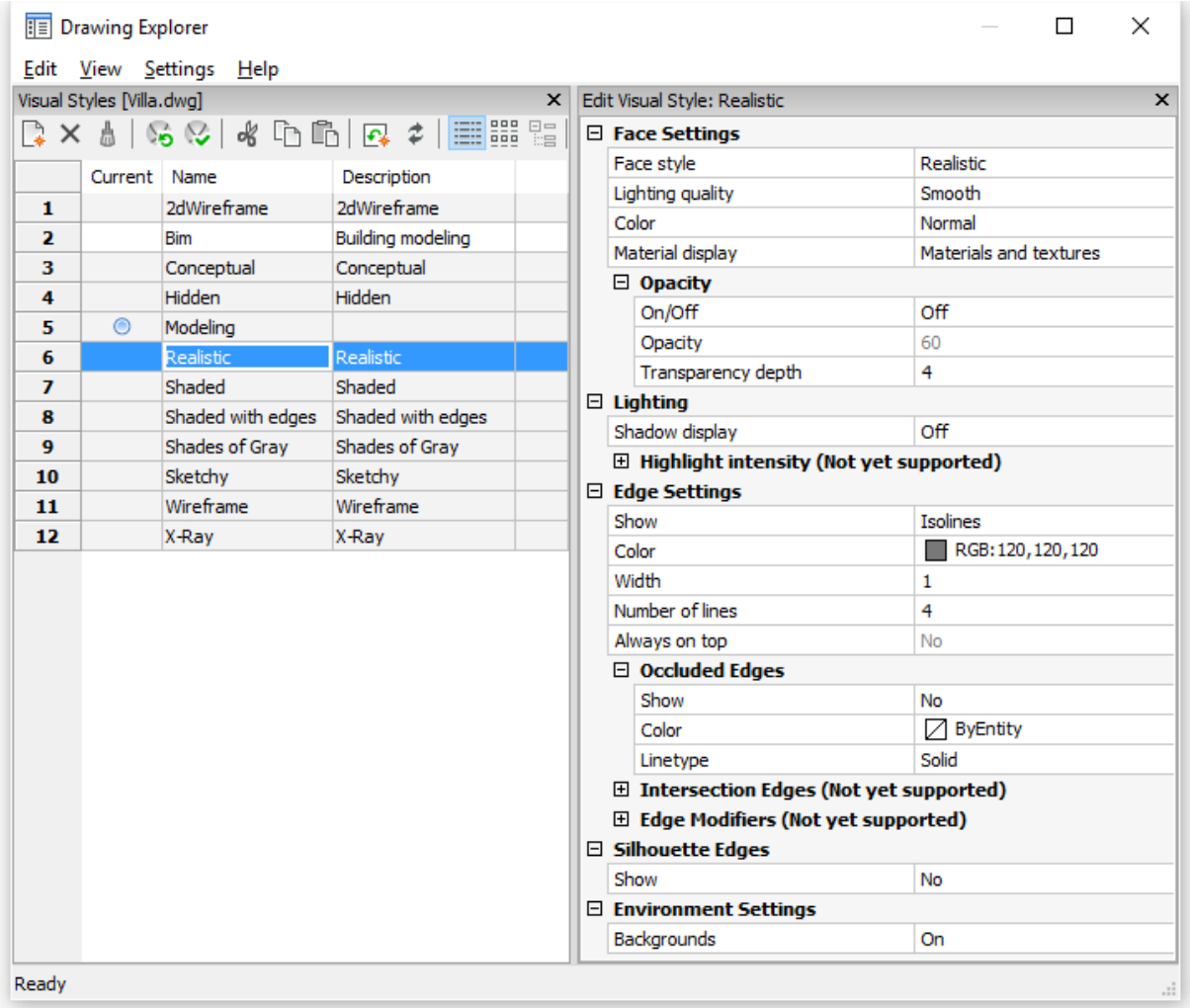
Creates and controls visual style definitions in the Drawing Explorer.

Accessing the Command

command bar: *visualstyles*
menu bar: *Tools | Drawing Explorer | Visual Styles...*

: ***visualstyles***

Displays a dialog box:



Command Options

Option	Description
New	Creates a new visual style.
Delete	Deletes the selected visual style. The predefined visual styles such as 2dWireframe, 3D Hidden, 3dWireframe, Conceptual, Realistic, ... are hard-coded and cannot be deleted or renamed.
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. *Mechanical Drafting* and *Mechanical Drafting Section* visual styles are used by the BmGenDraft and BmGenSection commands

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. Note: high anti-alias values incur a high calculation cost.

Related Commands

AniPath - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
Render - generates a photorealistic rendering using materials and lights.
Shademode - sets the shading style for 3D drawings.
-VisualStyles - manages visual style definitions at the command line
VsCurrent - sets the visual style at the command line.

Explorer - Opens the Drawing Explorer dialog box.

-VisualStyles

Manages visual style definitions at the command line.

Accessing the Command

command bar: *-visualstyles*

: ***-visualstyles***
Prompts you in the command bar:

Visual styles: set_Current/Saveas/Rename/Delete/?: ***(Enter an option.)***

Command Options

Option	Description
Set Current	Prompts you for a visual style to be set current: Enter visual style Wireframe/Hidden/Realistic/Conceptual/Shaded/shaded_with_Edges/shades_of_Grey/SKetchy/X-ray/<Other>: Do one of the following: • Type the capitalized letter of one of the default visual styles. • Press Enter to set a user-defined visual style. Prompts you: **Enter a visual style name or [?]:** Type a visual style name or hit the *?* key to see a list of the visual styles in the drawing.
Saveas	Saves the current visual style with a new name. Prompts you: Enter new visual style name or [?]: Type a name hit the *?* key to see a list of the visual styles in the drawing.
Rename	Renames a visual style. Prompts you: Enter visual style to rename or [?]: Type the name of an existing visual style or hit the *?* key to see a list of the visual styles in the drawing. **Enter new visual style name:** Type a new name.
Delete	Deletes a visual style. Prompts you: Enter visual style to delete or [?]: Type the name of an existing visual style or hit the *?* key to see a list of the visual styles in the drawing. You cannot delete default visual styles or the current visual style.
?	Lists the visual styles in the drawing. Press the *F2* function key to open the *Prompt History* window.

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. Note: high anti-alias values incur a high calculation cost.

Related Commands

AniPath - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
Render - generates a photorealistic rendering using materials and lights.
Shademode - sets the shading style for 3D drawings.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.
VsCurrent - sets the visual style at the command line.

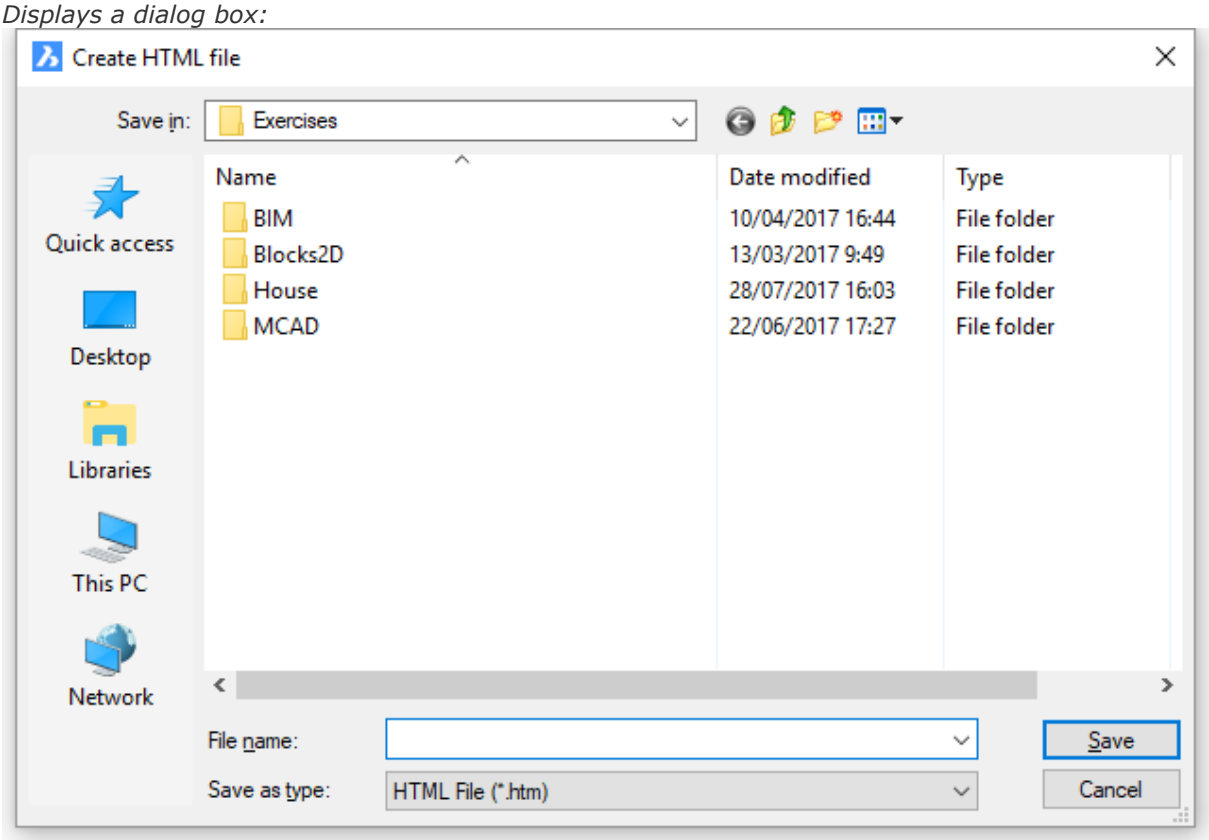
VmlOut

Exports drawings in VML format embedded in Web pages (short for "vector markup language").

Accessing the Command

command bar: *vmlout*

: ***vmlout***
Displays a dialog box:



Enter a file name, and then click Save**.**

The VML data is saved in an HTML file. To view the image, open in Internet Explorer with the VML plug-in installed.

Command Options

Option	Description
File name	Specifies the name of the file.
Save as type	Saves the VML data in an HTML file. HTML is used to display Web pages (short for "hyper text markup language").
Save	Saves the file.

Related Commands

BmpOut - exports drawings in BMP format.
Export - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats.
Import - imports DXF files.
MSlide - exports drawings in WMF, EMF, and SLD formats.
VSlide - displays files in WMF, EMF, and SLD formats.
WmfOut - exports drawings in WMF, EMF, and SLD formats.

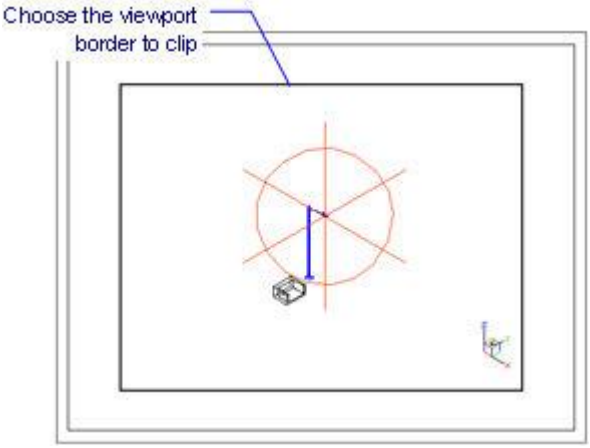
VpClip

Clips viewports in layouts (short for "view port clipping").

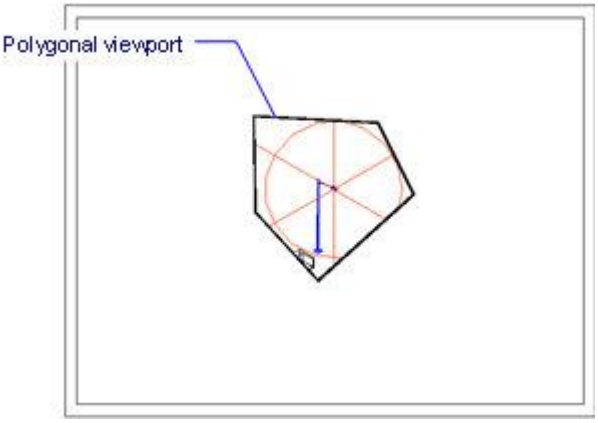
Accessing the Command

command bar: *vpclip*

: ***vpclip***
Prompts you in the command bar:
Select viewport to clip: **(Choose a viewport border.)**

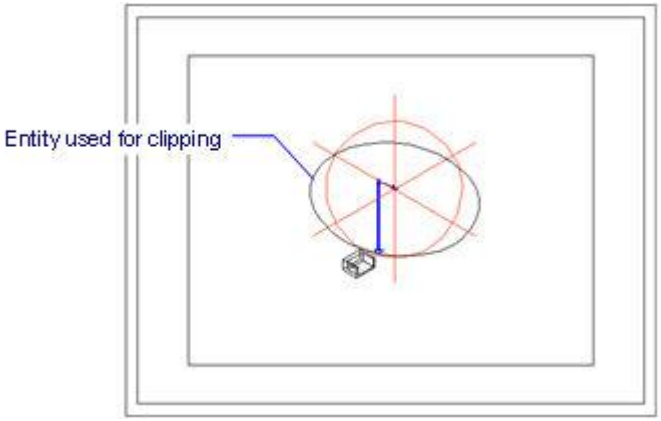
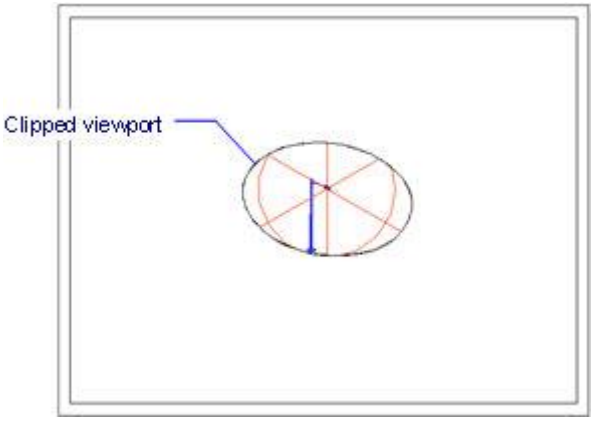


Select clipping entity or [Polygonal]: **(Choose a closed object, or type P.)**
Polygonal options:



This command can only be used in a layout tab (paper space).
Polygonal viewports must have at least three vertices.
Only closed entities, such as circles and closed polylines, can be used for viewport boundaries.

Command Options

Option	Description
Select viewport to clip	Chooses the viewport to clip. Clipped viewports can be reclipped. Use the **Delete** option to remove clipping.
Select clipping entity	Converts a closed object into the viewport boundary.  The entity must be drawn in paper space. 
Polygonal	Draws a polygonal viewport border made of straight and arc segments; prompts you: Specify start point - Pick a point. Set next point or [**draw Arcs**/**Distance**] - Pick a point or choose an option. Set next point or [draw Arcs/Distance/**Follow**/**Undo**] - Pick a point or choose an option. Set next point or [draw Arcs/**Close**/Distance/Follow/Undo] - Pick a point or choose an option or press Enter to automatically close the polygon.
Arc	Draws an arc segment; prompts you: Set end of arc or [draw Lines/Angle/CEnter/CLose/Direction/Radius/Second point/Undo] - Pick a point or enter an option. See **Arc** command.

Distance	Specifies the distance and angle for the next segment; prompts you: Distance for segment - enter a distance or show by picking a point. Angle of segment - enter an angle or show by picking a point.
Follow	Draws the next segment at the same angle as the previous segment. Length of line - specifies the length of the segment.
Undo	Undoes the last segment.
Close	Closes the polygon.
Delete	Deletes the clipping boundary, and returns the original rectangular viewport. (This option appears only when you select a clipped viewport.)

Related Commands

XClip - clips externally-referenced drawings.
ImageClip - clips images.
MView - creates viewports in paper space.
VPorts - creates viewports in model space.
MSpace - switches to model space.
PSpace - switches to paper space.

VpLayer

Changes the properties of layers in the current paperspace viewport (short for "view port layer").

Accessing the Command

command bar: *vplayer*
alias: *vl*

: ***vplayer***

This command operates only in layout mode.

In model space, this command responds, "This command can only be used in a Layout Tab ('Paper Space')."

Prompts you in the command bar:
[List frozen layers/Color/LineType/LineWeight/TRansparency/Freeze layers/Thaw layers/Reset layers/New frozen layers/Default visibility setting]: (Enter an option, or press Enter to leave properties unchanged.)

Command Options

Option	Description
List frozen layers	Reports whether layers are frozen in the selected viewport; prompts you: **Select a viewpoint** - click a viewport border; responds with the following report: The following layers are frozen in viewport 2: LayerName When no layers are frozen, reports: No layers are frozen in viewport.
Color	Sets the VP Color property. Prompts you: **New truecolor: Red,Green,Blue** - type the RGB code (e.g. 123,123,123) **Select layer name(s) to override color -** type the layer names separated by commas. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - choose an option or press Enter. • **All**: apply to all viewports in the current layout. • **Select**: Allow to select the viewports; prompts you: **Select a viewport:** click a viewport Entities in set: 1 **Select a viewport:** click more viewports or press Enter. • **Current**: apply to the current viewport only. • **Except current:** apply to all viewports, except the current viewport.
Line Type	Sets the VP Linetype property. Prompts you: **Loaded linetype name or [? to list] <Continuous>** - type a linetype name or press Enter, or choose an options. • ? to list - displays a list of available linetypes in the command bar. **Select layer name(s) to override linetype** - type the layer names separated by commas.

	Specify viewport(s) [All/Select/Current/eXcept current] <Current> - choose an option or press Enter. • **All**: apply to all viewports in the current layout. • **Select**: Allow to select the viewports; prompts you: **Select a viewport:** click a viewport Entities in set: 1 **Select a viewport:** click more viewports or press Enter. • **Current**: apply to the current viewport only. • **Except current:** apply to all viewports, except the current viewport.
Line Weight	Sets the VP Lineweight property. Prompts you: **Enter lineweight (0.00 mm - 2.11 mm)** - type a lineweight. **Select layer name(s) to override lineweight** - type the layer names separated by commas. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - choose an option or press Enter. • **All**: apply to all viewports in the current layout. • **Select**: Allow to select the viewports; prompts you: **Select a viewport:** click a viewport Entities in set: 1 **Select a viewport:** click more viewports or press Enter. • **Current**: apply to the current viewport only. • **Except current:** apply to all viewports, except the current viewport.
Transparency	Sets the VP Transparence property. Prompts you: **Enter transparency (0 - 90)** - type a value. **Select layer name(s) to override transparency** - type the layer names separated by commas. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - choose an option or press Enter. • **All**: apply to all viewports in the current layout. • **Select**: Allow to select the viewports; prompts you: **Select a viewport:** click a viewport Entities in set: 1 **Select a viewport:** click more viewports or press Enter. • **Current**: apply to the current viewport only. • **Except current:** apply to all viewports, except the current viewport.
Freeze layers	Freezes layers in viewports; prompts you: **Select the layer(s) to freeze** - enter the names of layers to freeze. Separate multiple layer names with commas.
Thaw layers	Thaws layers in viewports; prompts you: **Select the layer(s) to reset** - enter the names of layers to thaw. Separate multiple layer names with commas.
Reset layers	Resets frozen and thawed layers to their original settings; prompts you: **Select the layer(s) to reset** - enter the names of layers to reset. Separate multiple layer names with commas.

New frozen layers	Creates new layers that are frozen by default; prompts you: **New frozen layer name(s)** - enter the names of layers to create and freeze. Separate multiple layer names with commas.
Default visibility settings	Changes the default freeze/thaw setting for layers; prompts you: **Layer name(s)** - enter one or more layer names; separate multiple layer names with commas. **Setting Frozen/<Thawed>** - type F or T. • **Frozen** - changes the layer's default property to frozen. • **Thawed** - changes the layer's default property to thawed.

Related Commands

-Layer - creates layers and changes their properties in the command bar.
Layer - creates layers, changes their properties, and removes layers through the Drawing Explorer.
LayersPanelOpen - opens the Layers panel, which allows to edit the layers in the current drawing, apply layer states and layer filters.

VPoint

Changes the 3D viewpoint.

Accessing the Command

command bar: *vpoint*
transparent: *'vpoint*
aliases: *viewpoint, -vp, -viewpoint, -vpoint*

: ***vpoint***
Prompts you in the command bar:
Set view point or [Rotate/PLan view/PErspective] <0, 0, 1>: (Enter x,y,z coordinates, an option, or press Enter for dialog box.)

Command Options

Option	Description
Set view point	Specifies the viewpoint by x,y,z coordinates. E.g. to set a front left isometric viewpoint, enter -1,-1,1.
Rotate	Changes the viewpoint by angles; prompts you: Angle in XY plane from X axis <45> - specify an angle, or pick two points. Angle from the XY plane <60> - specify an angle, or pick two points.
Plan view	Displays the plan view of the current UCS. See the Plan command.
Perspective	Allows to set the *Perspective* property of the viewport; prompts you: Perspective [perspective ON/perspective OFf] <Perspective off> - choose an option: • ON: Sets the Perspective property of the viewport on. • OFF: Sets the Perspective property of the viewport off.

Look From control

The Look From control allows to select a number of preset views such as orthographic views and isometric views.

Dialog Box Options

See DdVpoint command.

Related Commands

DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
DView - changes the 3D viewpoint interactively, and turns on perspective mode.
RtRot - rotates 3D drawings in real-time

VPorts

Creates one or more viewports in model space (short for "viewports").

Accessing the Command

command bar: *vports*
aliases: *viewports, vport, vw, -vports*
menu bar: *View | Viewports*
toolbar: *View* | and *Layout* |

: ***vports***

(This command runs only in model space. To create viewports in layout tabs, use the **MView** command.)

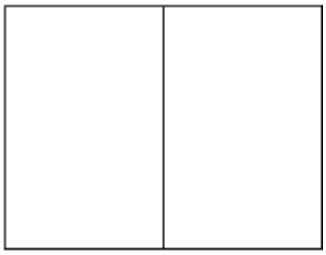
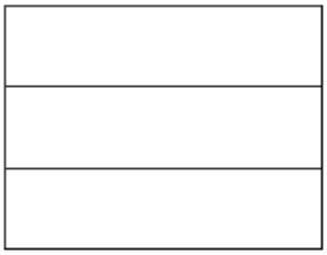
Prompts you in the command bar:

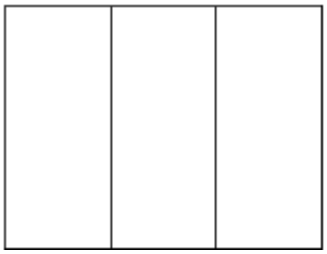
Viewports [? to list/Save/Restore/Delete/Join/SIngle/create 2 viewports/create 3 viewports/create 4 viewports] <Create 3 viewports>: (Enter an option or press Enter to create 3 viewports.)

The viewports allow you to see multiple viewpoints of the same drawings.

Command Options

Option	Description
? to list	Lists the names of saved viewports. A typical display looks like this: **Viewport Name LL Corner UR Corner** **----------** ***Active 0.312500,0.000000 0.343750,0.125000** **SPLIT 0.343750,0.125000 0.375000,0.250000** LL is short for lower-left corner; UR is short for upper-right corner. Press **F2** to view the list in the prompt history window.
Save	Saves the current viewport arrangement by name; prompts you: Viewport configuration to save - specifies the name of the viewport configuration. If the name already exists, prompts you: A viewport configuration with that name exists, Overwrite?(Yes/No)? - type Y or N. • Yes - replaces the viewport configuration with the new one. • No - repeats the prompt, 'Viewport configuration to save'; enter a different name.
Restore	Restores a named viewport configuration; prompts you: Viewport configuration to restore - enter the name of the viewport configuration.
Delete	Erases a named viewport configuration from the drawing; prompts you: Viewport configuration to delete - specify the name of a viewport configuration. You cannot delete more than one viewport configuration at a time.
SIngle	Creates a single viewport of the current viewport. All other viewports are removed.

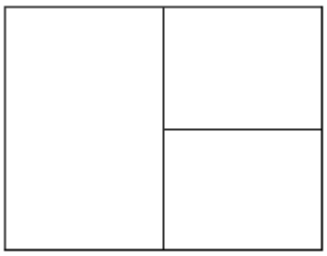
Join	Joins two or more viewports into a single viewport; prompts you: Select dominant viewport <Current> - choose the viewport that will be retained. Select viewport to join - choose the viewport that will unified with the selected viewport. When the two viewports form a non-rectangular shapes (such as an L or T shape), BricsCAD complains, "The selected viewports do not form a rectangle." The 'Select viewport to join' prompt repeats.
2	Creates two rectangular viewports; prompts you: Two viewports [Horizontal/Vertical] <Vertical> - type H or V. • Horizontal - draws two horizontal viewports. • Vertical - draws two vertical viewports.  The current viewport is split into two.
3	Draws three rectangular viewports; prompts you: **Three viewports [Horizontal/Vertical/Above/Below/Left/Right] <Right>** - enter an option. • Horizontal - draws three horizontal viewports.  • Vertical - draws three vertical viewports.



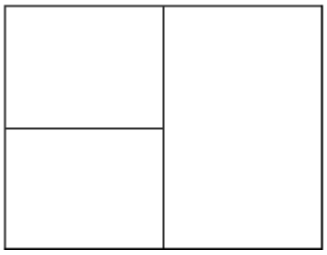
- Above - draws one viewport above two side-by-side viewports.

- Below - draws two side-by-side viewports above one viewport.

- Left - draws one viewport to the left of two stacked viewports.



- Right - draws one viewport to the right of two stacked viewports.



The current viewport is split into three viewports.

4	Draws four rectangular viewports. The current viewport is split into four viewports.

Procedures

Creating viewports in model space
Drawing in multiple viewports
Joining adjacent viewports

Related Commands

VpClip - clips viewports.
MView - creates viewports in paper space.
MSpace - switches to model space.
PSpace - switches to paper space.
RedrawAll - redraws all viewports.
RegenAll - regenerates the drawing in all viewports.
Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

-VPorts

Creates one or more viewports in model space (short for "viewports").

Accessing the Command

command bar: *vports*
aliases: *viewports, vport, vw, -vports*
menu bar: *View | Viewports*
toolbar: *View* | and *Layout* |

: ***vports***

(This command runs only in model space. To create viewports in layout tabs, use the **MView** command.)

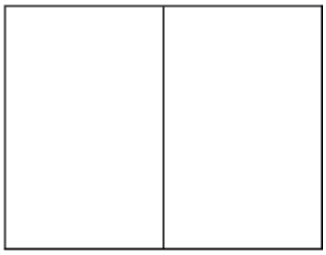
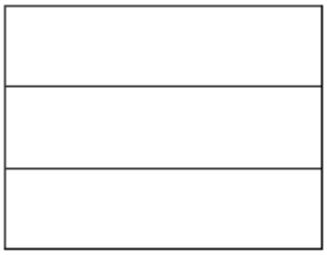
Prompts you in the command bar:

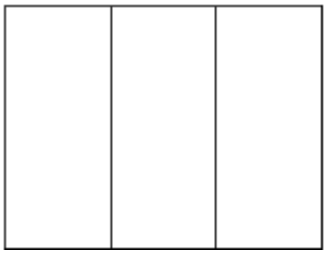
Viewports [? to list/Save/Restore/Delete/Join/SIngle/create 2 viewports/create 3 viewports/create 4 viewports] <Create 3 viewports>: (Enter an option or press Enter to create 3 viewports.)

The viewports allow you to see multiple viewpoints of the same drawings.

Command Options

Option	Description
? to list	Lists the names of saved viewports. A typical display looks like this: **Viewport Name LL Corner UR Corner** **----------** ***Active 0.312500,0.000000 0.343750,0.125000** **SPLIT 0.343750,0.125000 0.375000,0.250000** LL is short for lower-left corner; UR is short for upper-right corner. Press **F2** to view the list in the prompt history window.
Save	Saves the current viewport arrangement by name; prompts you: Viewport configuration to save - specifies the name of the viewport configuration. If the name already exists, prompts you: A viewport configuration with that name exists, Overwrite?(Yes/No)? - type Y or N. • Yes - replaces the viewport configuration with the new one. • No - repeats the prompt, 'Viewport configuration to save'; enter a different name.
Restore	Restores a named viewport configuration; prompts you: Viewport configuration to restore - enter the name of the viewport configuration.
Delete	Erases a named viewport configuration from the drawing; prompts you: Viewport configuration to delete - specify the name of a viewport configuration. You cannot delete more than one viewport configuration at a time.
SIngle	Creates a single viewport of the current viewport. All other viewports are removed.

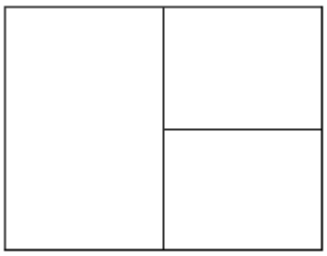
Join	Joins two or more viewports into a single viewport; prompts you: Select dominant viewport <Current> - choose the viewport that will be retained. Select viewport to join - choose the viewport that will unified with the selected viewport. When the two viewports form a non-rectangular shapes (such as an L or T shape), BricsCAD complains, "The selected viewports do not form a rectangle." The 'Select viewport to join' prompt repeats.
2	Creates two rectangular viewports; prompts you: Two viewports [Horizontal/Vertical] <Vertical> - type H or V. • Horizontal - draws two horizontal viewports. • Vertical - draws two vertical viewports.  The current viewport is split into two.
3	Draws three rectangular viewports; prompts you: **Three viewports [Horizontal/Vertical/Above/Below/Left/Right] <Right>** - enter an option. • Horizontal - draws three horizontal viewports.  • Vertical - draws three vertical viewports.



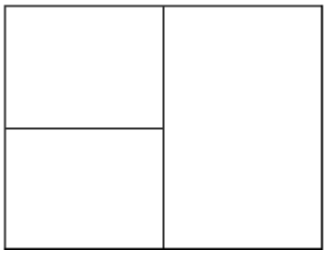
- Above - draws one viewport above two side-by-side viewports.

- Below - draws two side-by-side viewports above one viewport.

- Left - draws one viewport to the left of two stacked viewports.



- Right - draws one viewport to the right of two stacked viewports.



The current viewport is split into three viewports.

4	Draws four rectangular viewports. The current viewport is split into four viewports.

Procedures

Creating viewports in model space
Drawing in multiple viewports
Joining adjacent viewports

Related Commands

VpClip - clips viewports.
MView - creates viewports in paper space.
MSpace - switches to model space.
PSpace - switches to paper space.
RedrawAll - redraws all viewports.
RegenAll - regenerates the drawing in all viewports.
Zoom - utilizes the XP option to specify the zoom factor in paper space relative to model space.

VsCurrent

Sets the visual style at the command line.

Accessing the Command

command bar: *vscurrent*

: ***vscurrent***
Prompts you in the command bar:
Enter visual style [2dwireframe/Wireframe/Hidden/Realistic/Conceptual/Shaded/shaded with Edges/shades of Grey/SKetchy/X-ray/Other/cUrrent] <Modeling>: (Enter an option.)

Command Options

Option	Description
Other	Allows to choose a user defined visual style. Enter a visual style name or [?]: • Type the name of a visual style, then press Enter. • (option) Type ? + Enter to list all available visual styles. Type *F2* to open the *Prompt History* window.

Related Commands

-VisualStyles - manages visual style definitions at the command line.
VisualStyles - creates and controls visual style definitions in the Drawing Explorer.

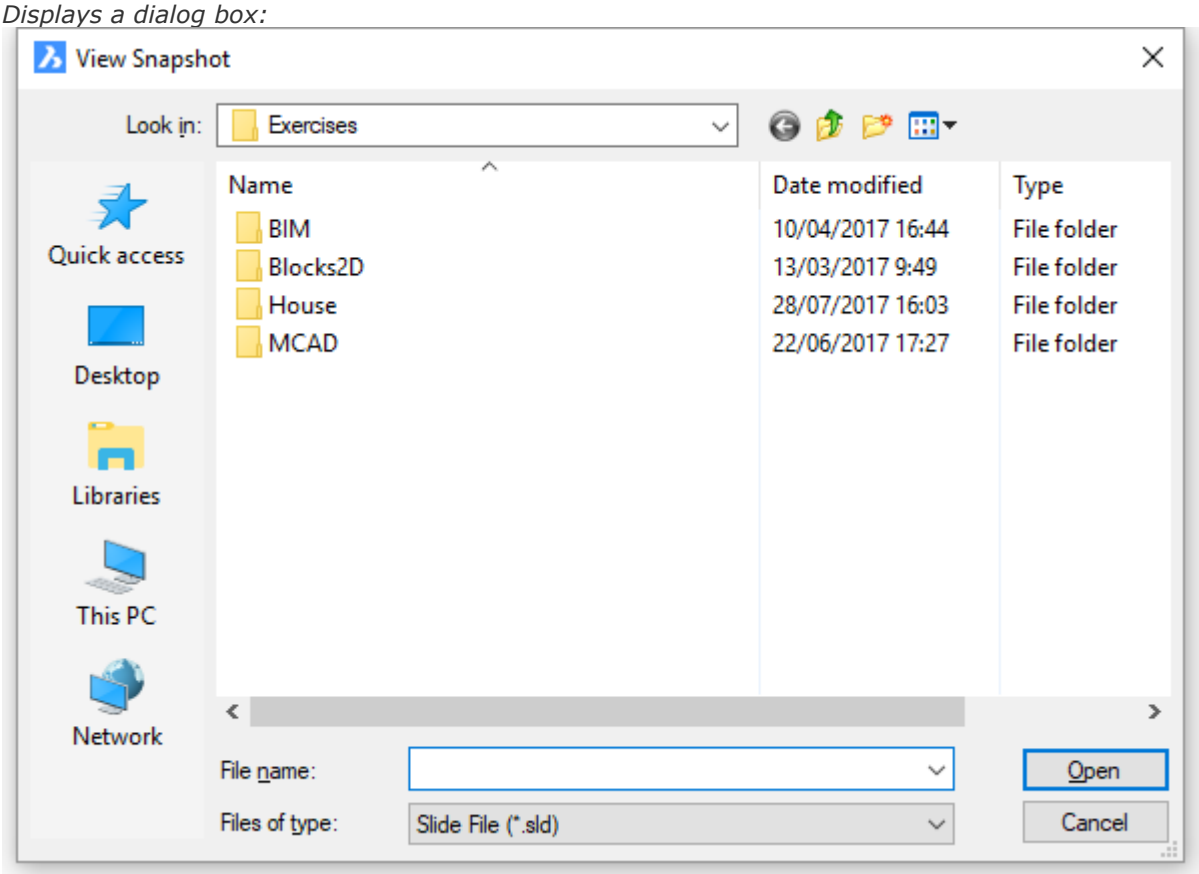
VSlide

Displays images saved as SLD or WMF files (short for "view slide").
WMF files are not supported on the Linux platform.

Accessing the Command

command bar: *vslide*
aliases: *vs, vsnapshot*
menu bar: *Tools | View Snapshot*
toolbar: *Tools |*

: ***vslide***
Displays a dialog box:



Choose a file, and then click Open.

To remove the slide from the viewport, use the **Redraw** command.

Command Options

Option	Description
File name	Specifies the name of the slide file.
Files of type	Selects the file type of the slide file: • Slide File (*.sld) is the proprietary graphics format used by AutoCAD for slides. • Slide Library (*.slb) contains a collection of slides (.sld) files; used to display slideshows of CAD designs. • Enhanced Metafile (*.emf) is the enhanced WMF format. • Windows Metafile (*.wmf) is the hybrid raster-vector format developed by Microsoft.
Open	Displays the slide in the current viewport. Use the **Redraw** command to remove the slide.

Related Commands

MSlide - saves drawings in SLD, EMF, and WMF formats.
Redraw - removes the slide from the viewport.

VBA

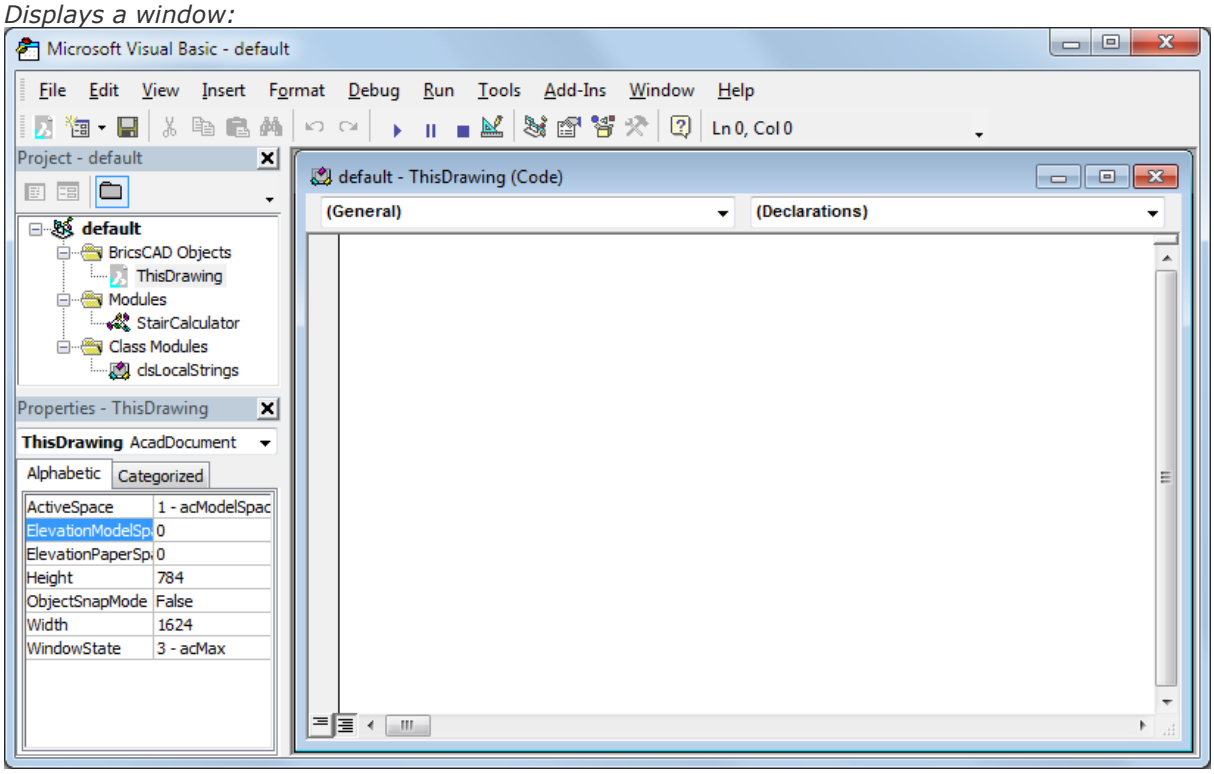
VbaIde

Opens the VBA editing window (short for "visual basic for applications integrated development environment").

Accessing the Command

command bar: *vbaide*
shortcut: *Alt+F11*
alias: *vba*
menu bar: *Tools | VBA | Visual Basic for Applications*
toolbar: *Tools |*

: ***vbaide***
Displays a window:



Command Options

See the help files provided by Microsoft for Visual Basic.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaUnload - unloads VBA projects.

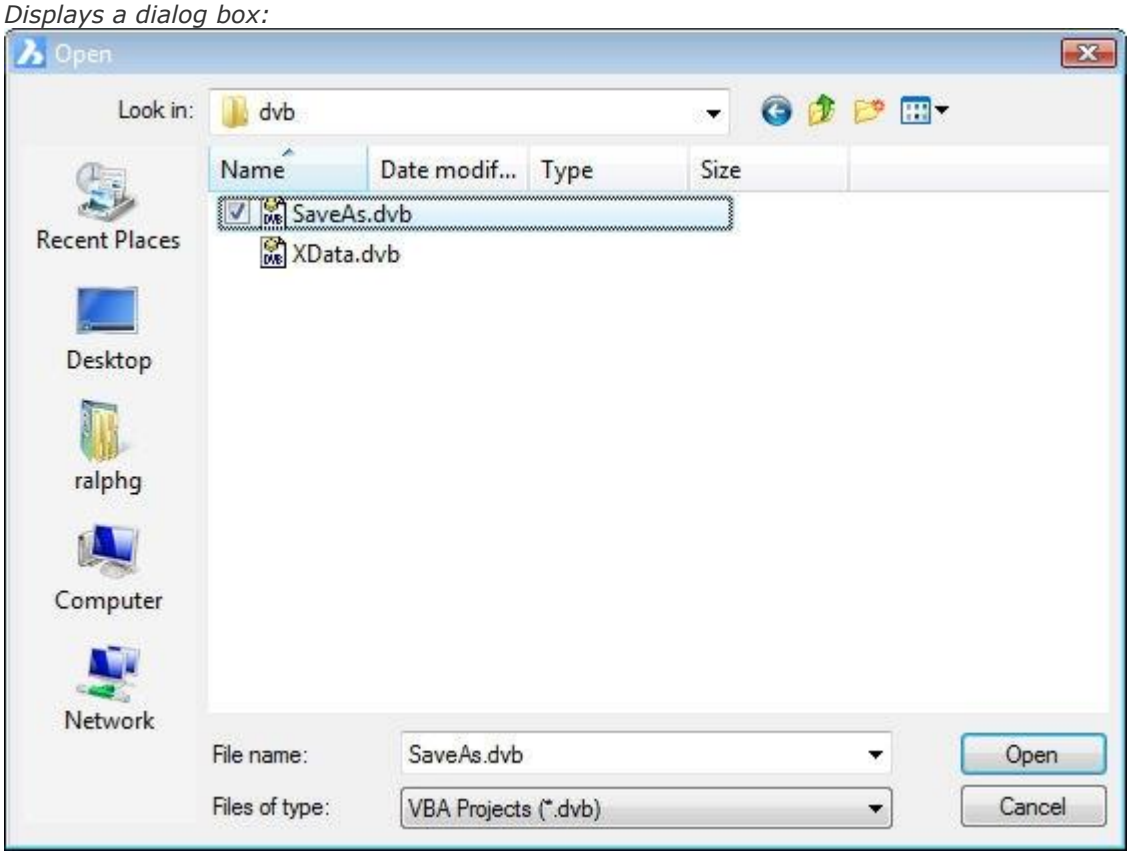
VbaLoad

Loads VBA projects through a dialog box (short for "visual basic for applications load").

Accessing the Command

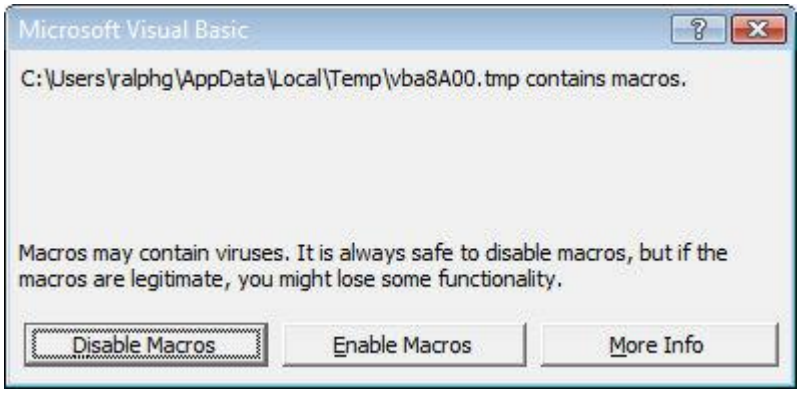
command bar: *vbaload*
menu bar: *Tools | VBA | Load Project*

: ***vbaload***
Displays a dialog box:



Choose a DVB file, and then click Open**.**

BricsCAD warns about the potential hazard of macros:



Click Enable Macros; see the **Security** command for details.

To run the macros in the project, use the **VbaRun** command.

Command Options

Option	Description
File name	Specifies the name of the DVB file. (DVB files hold VBA projects.)
Open	Opens the DVB file.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
-VbaLoad - loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

-VbaLoad

Loads VBA projects in the command bar (short for "visual basic for applications load").

Accessing the Command

command bar: *-vbaload*

: ***-vbaload***
Prompts you in the command bar:

VBA Project Name: (Enter the path and name of a DVB file.)

This command will not load VBA projects that are embedded in drawings, because opening a drawing loads its associated project automatically.

When you specify the name of an already-loaded DVB file, BricsCAD reports, "Project is already loaded."

Command Options

Option	Description
VBA Project Name	Names the DVB file. DVA are VBA project files, which contain one or more VBA macros.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
VbaLoad loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

VbaUnload

Unloads VBA projects (short for "visual basic for applications unload").
This command is not available in the 64 bit version.

Accessing the Command

command bar: *vbaunload*

: ***vbaunload***
Prompts you in the command bar:

VBA Project Name: (Enter the name of a DVB project.)

To unload the current project, press Enter at the 'VBA Project Name" prompt.

Command Options

Option	Description
VBA Project Name	Names the DVB file. DVA files are VBA project files, which contain one or more VBA macros.

Related Commands

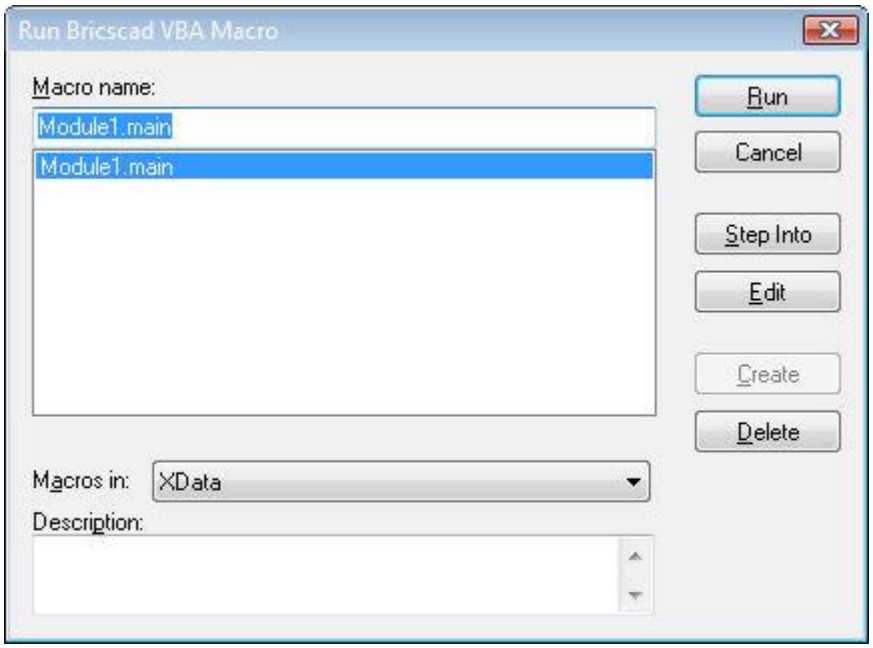
AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.

VbaRun

Runs, creates, edits, and deletes VBA macros through a dialog box (short for "visual basic for applications run").

Accessing the Command

command bar: *vbarun*
menu bar: *Tools | VBA | Macros*
: ***vbarun***
Displays a dialog box:



Choose the name of a macro, and then click Run.

Command Options

Option	Description
Macro name	Lists the names of macros found in the project file. Choose one to run.
Macros in	Lists the names of projects loaded into the current drawing. Choose one from the droplist.
Description	Describes the macro. You can edit this text.
Run	Runs the selected macro.
Cancel	Closes the dialog box.
Step Into	Opens the integrated development environment window. The macro is in debug mode, which runs one line of code at a time. See **VbaIde** command.
Edit	Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code. See **VbaIde** command.

Create	Creates new macros; follow these steps: 1. In the Macro Name field, enter a name for the new macro 2. Click Create. 3. Notice the integrated development environment window. Enter VBA code. See **VbaIde** command.
Delete	Deletes the macro; prompts you: Are you sure you want to delete the macro - enter Y or N. • Yes - erases the macro. • No - does not erase the macro.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaLoad and -VbaLoad - loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.

-VbaRun

Runs VBA macros in the command bar (short for "visual basic for applications run").

Accessing the Command

command bar: *-vbarun*
: ***-vbarun***
Prompts you in the command bar:
Macro name: (Enter the name of a VBA macro.)

- If the macro is part of a project and a module, enter its name using this format:

project.module.macro

- If the DVB file is not loaded into BricsCAD, enter the name of the macro using this format:

project.dvb!project.module.macro

This command is meant for use by macros and programs.

Command Options

Option	Description
Macro name	Names the VBA macro to be run.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
VbaLoad and -VbaLoad - loads VBA projects.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

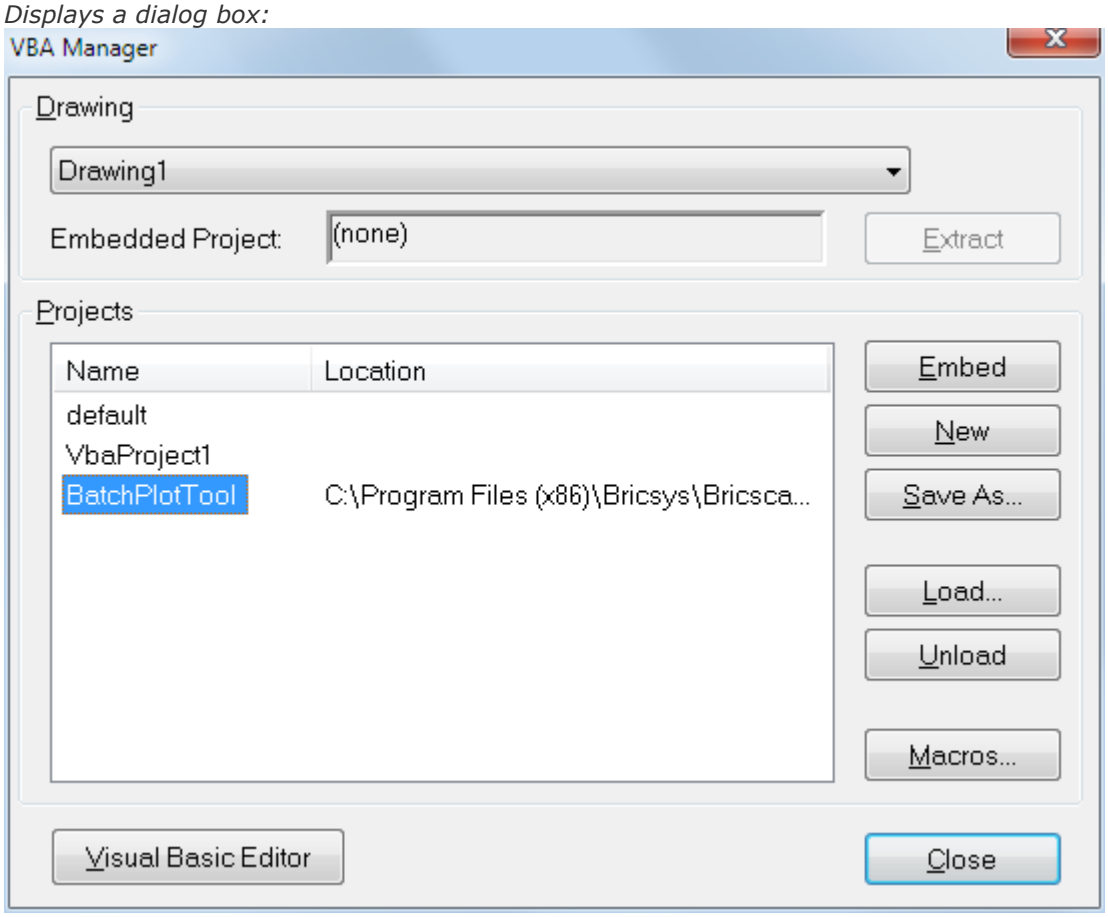
VbaMan

Manages VBA projects (short for "visual basic for applications manager").

Accessing the Command

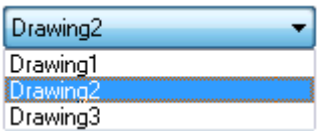
command bar: *vbaman*
menu bar: *Tools | VBA | Project Manager*

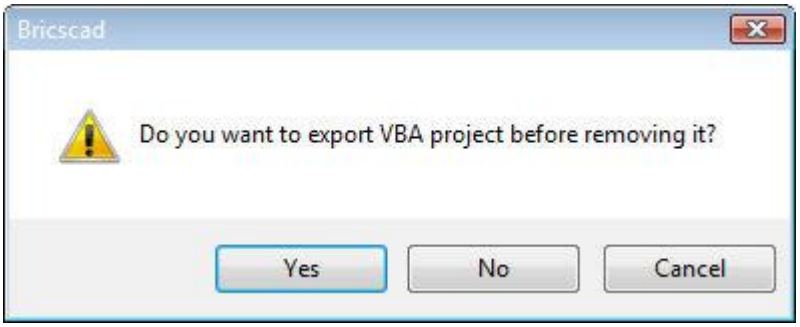
: ***vbaman***
Displays a dialog box:



Choose options, and then click Close.

Command Options

Option	Description
Drawing	Lists the names of drawings currently open in BricsCAD:  Choose a drawing name from the droplist; the embedded project is listed in the Embedded Project text box, if any.
Embedded Project	Lists the name of the embedded project, if any.

Projects	
Name	Lists the names of projects loaded into BricsCAD at this time.
Location	Lists the drive and folder names of DVB files.
Extract	Removes the project from the current drawing. Displays dialog box:  • **Yes** - displays the Save As dialog box: 1. Name the project. 2. Click **Save**. • **No** - removes the project from the drawing without saving it as a DVB file. • **Cancel** - cancels the command; the project remains in the drawing.
Embed	Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the Save As dialog box. Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the **Extract** option to remove the current project.
New	Creates a new VBA project; gives the generic name *VbaProject*. Use the **Save As** option to rename and save the DVB project file.
Save As	Saves the selected project by another name. Displays the Save As dialog box: 1. Enter a file name. 2. Click **Save**.
Load	Loads DVB project files. Displays the Open dialog box: 1. Choose a DVB file. 2. Click **Open**. See **VbaLoad** command.
Unload	Unloads the selected project without a warning.
Macros	Displays the Run BricsCAD VBA Macro dialog box: 1. Choose the macro to run. 2. Click **Run**. See **VbaRun** command.
Visual Basic Editor	Opens the VBA integrated development environment. See **VbaIde** command.
Close	Closes the dialog box.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaIde - opens the VBA integrated development environment.
VbaUnload - unloads VBA projects.

VbaNew

Starts a new VBA project.

Accessing the Command

command bar: *vbanew*
menu bar: *Tools | VBA | New Project*

: ***vbanew***

Command Options

No options.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
vbaide - opens the VBA editing window.
VbaRun and -VbaRun - runs VBA macros.
VbaMan - manages VBA projects.
VbaUnload - unloads VBA projects.

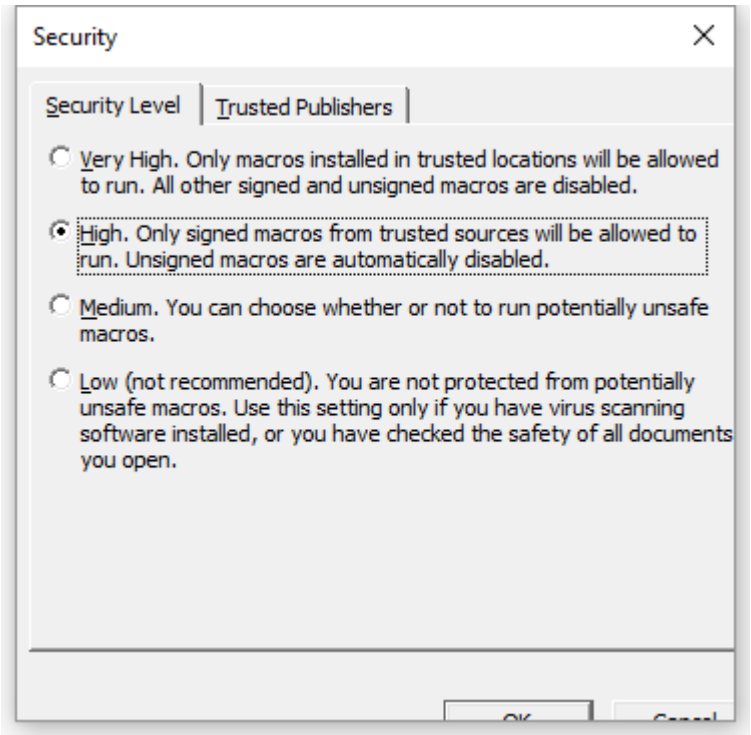
VbaSecurtiy

Depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

Accessing the Command

command bar: *vbasecurity*
menu bar: *Tools | VBA | Security*

: ***security***
Displays a dialog box:



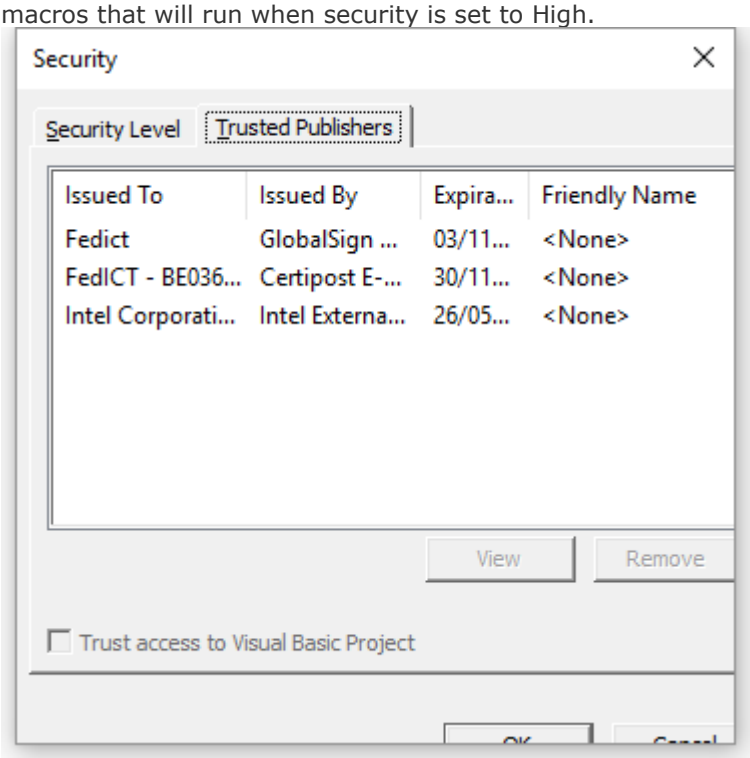
Choose a level of security, and then click **OK**.

This dialog box is necessary, because Microsoft's design of VBA applications allows use of destructive code, which can damage files on your computer.

VBA is short for Visual Basic for Applications; it is a programming language found in many Windows applications.

Command Options

Option	Description
Very High	Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.
High	Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.
Medium	BricsCAD displays a dialog box when a VBA macro runs. The dialog box allows you to choose whether or not to run the macro.
Low	All VBA macros are run.

Trusted Sources Option	Description
Trusted Sources	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.

Related Commands

AddInMan - loads and unloads VBA functions through a dialog box.
AppLoad - selects external routines to load when BricsCAD starts.
Netload - loads a .NET application.
Security - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.
VbaIde - opens the VBA integrated development environment.
VbaMan - manages VBA projects.
VbaNew - starts a new VBA project.
VbaRun and -VbaRun - runs VBA macros.
VbaUnload - unloads VBA projects.

W

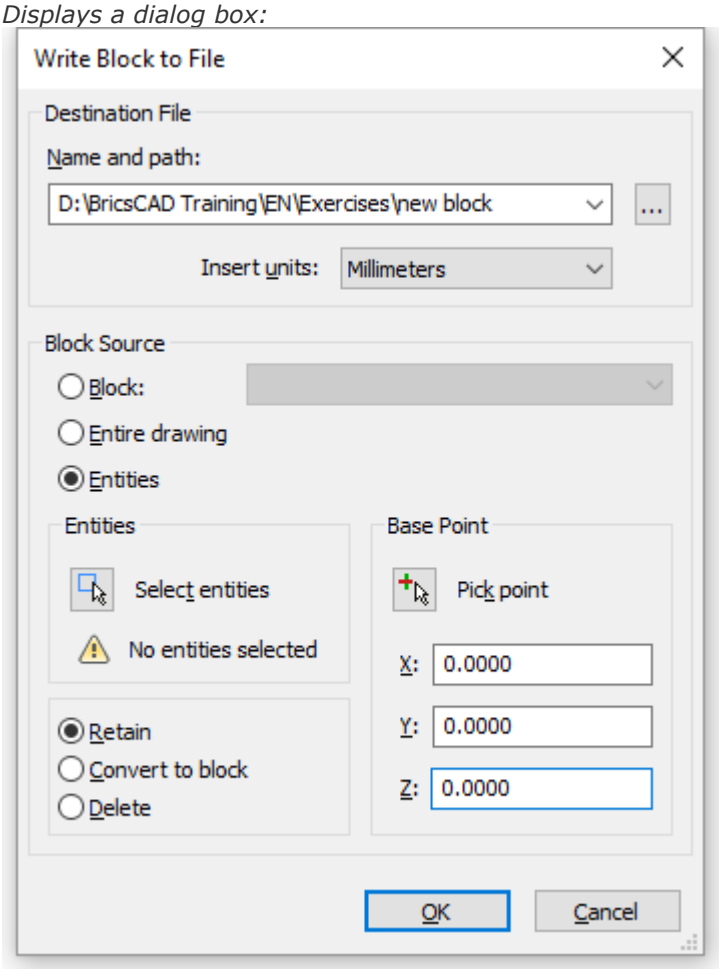
WBlock

Writes blocks and other drawing parts to files on disk through a dialog box (short for "write block").

Accessing the Command

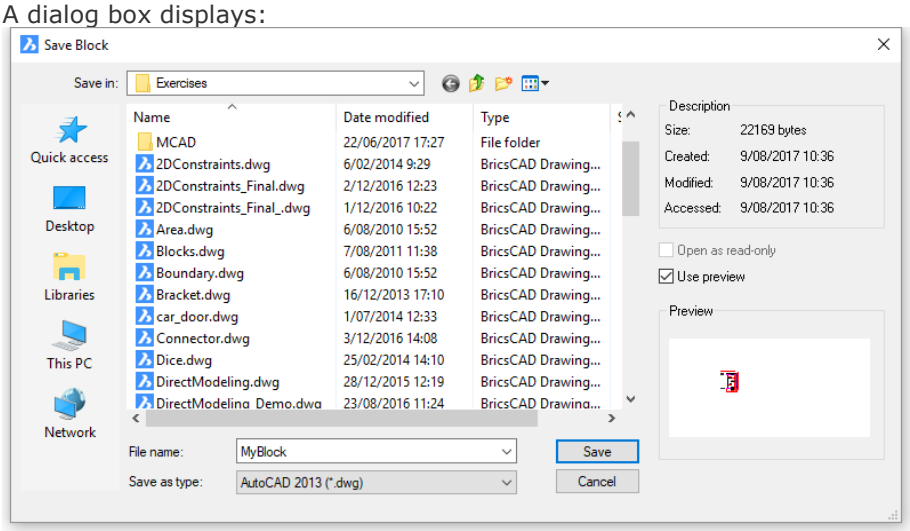
command bar: *wblock*
alias: *w*
menu bar: *Tools | Write Block*
toolbar: *Tools |*

: ***wblock***
Displays a dialog box:



Command Options

Option	Description

Name and Path	Click the *Browse* button (...).
	A dialog box displays:
	 
	
	• Save in - Select a folder
	• File name - Specifies the name of the file.
	• Save as type - Specifies the file type. Choose from DWG or DXF formats in the list.
	• Use Preview - Toggles inclusion of the preview image in the file:
	  • Yes - includes a bitmap preview image.
	  • No - does not include the image.

Insert Units	Specifies the INSUNITS (insertion units) system variable of the drawing.
Block Source	Selects the geometry to save to a new file.
Block	Click the list button, then choose a block in the list
Entire drawing	Selects all geometry in the drawing.
Entities	Lets you select the entities to be saved to a new drawing. Click the *Select* button ( ). You are prompted to select entities. Press Enter or right click to stop selecting entities. • Retain - keeps the selected entities as is. • Convert to block - converts the selected entities to a block in the current drawing. • Delete - deletes the selected entities in the current drawing, thus moving the selection to a new drawing.
Base Point	Specifies the origin point of the new drawing. Do one of the following • Click the *Select* button ( ). You are prompted to select the base point in the drawing. • Type the coordinates of the base point in the X, Y and Z fields.

Procedures

To save a block to a separate file

Related Commands

-WBlock - writes blocks and other drawing parts to files on disk through the command bar.
Export - exports the drawing in a variety of raster and vector formats.
ExportLayout - exports all visible objects from the current layout to the model space of a new drawing.
Insert - inserts blocks and drawings through a dialog box.
-Insert - inserts blocks and drawings through the command bar.
MInsert - inserts blocks as a rectangular array.
SaveAs - saves drawings in DWG, DWF, and DXF formats for a variety of versions.

-WBlock

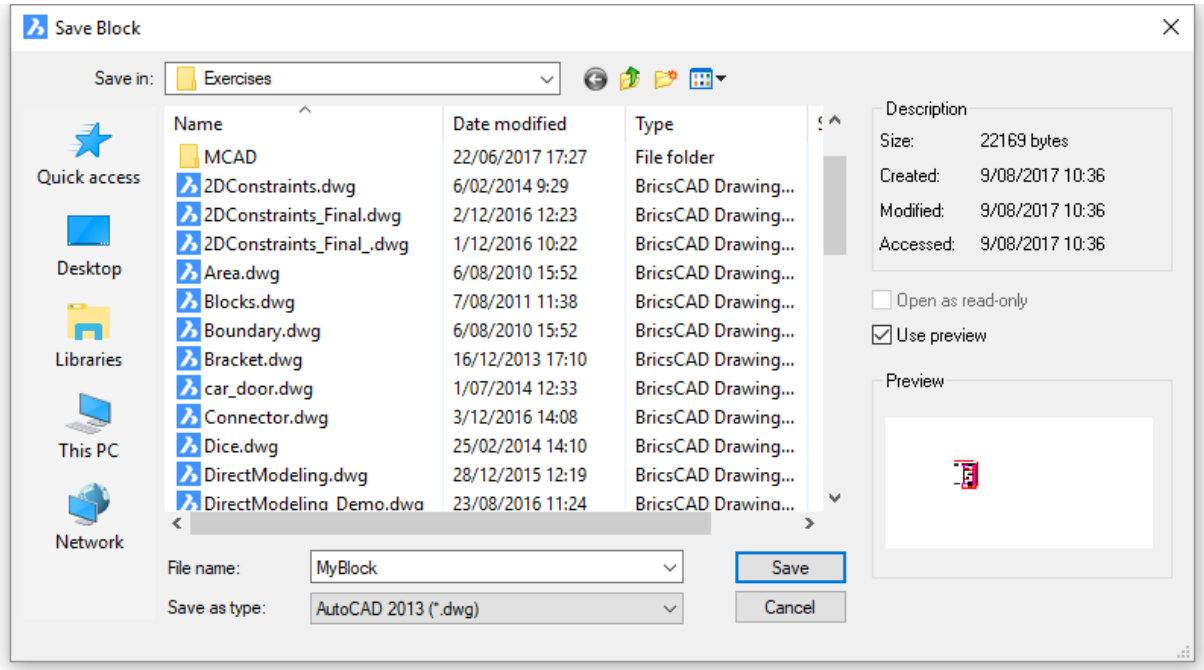
(short for "write block").

Accessing the Command

command bar: *-wblock*

: ***-wblock***

A File dialog displays:



Select a folder, then type a name in the File name field and press the Save button.

Prompts you in the command bar:
Block to save as WBlock or [& to include multiple blocks/* for whole drawing] <Select entities>: (Type the name of the block you want to write to a file or press Enter to select entites or enter an option.)

Command Options

Option	Description
Block to save as WBlock	Type the name of the block you want to write to a file.
Select entities	Press Enter. BricsCAD prompts you: **Insertion point for new block:** Specify the origin point of the drawing file to be created. **Select entities for block:** Select entities. **Entities in set: <number of entities>** **Select entities for block:** Select more entities or press Enter to stop and create the new drawing. The selected entities are deleted from the current drawing. Use the Oops command to restore the entities.

Multiple blocks	Allows to save multiple blocks to a drawing. Type & or M BricsCAD prompts you: **Blocks to save:** Type the names of the blocks separated by commas. The block definitions are saved to the new drawing.
Whole drawing	Saves the whole drawing. This procedure is equivalent to SaveAs. Type * or A All entities in the current drawing are saved to the new drawing.

Related Commands

WBlock - writes blocks and other drawing parts to files on disk through a dialog box.
Export - exports the drawing in a variety of raster and vector formats.
ExportLayout - exports all visible objects from the current layout to the model space of a new drawing.
Insert - inserts blocks and drawings through a dialog box.
-Insert - inserts blocks and drawings through the command bar.
MInsert - inserts blocks as a rectangular array.
SaveAs - saves drawings in DWG, DWF, and DXF formats for a variety of versions.

WCascade

Cascades windows (short for "window cascade").

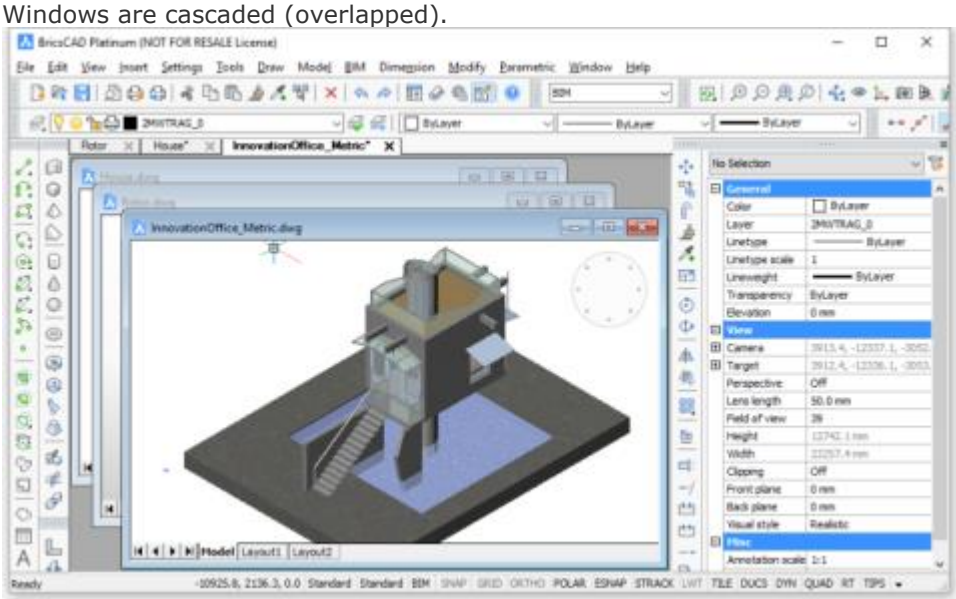
Accessing the Command

command bar: *wcascade*
menu bar: *Window | Cascade*

: ***wcascade***

Displays no prompt in the command bar.

Windows are cascaded (overlapped).



Command Options

No options.

Related Commands

SysWindows - arranges windows.
WClose - closes the current window.
WCloseAll - closes all windows.
WhTile - tiles windows horizontally.
WiArrange - arranges iconized windows.
WvTile - tiles windows vertically.

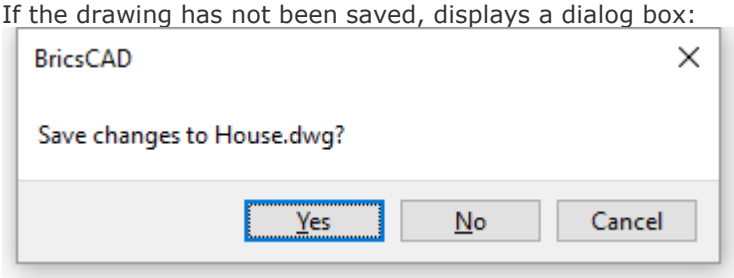
WClose

Closes the current window.

Accessing the Command

command bar: *wclose*
shortcut: *Ctrl+F4*
alias: *close*
menu bar: *Window | Close*

: ***wclose***
If the drawing has not been saved, displays a dialog box:



Click a button.

Command Options

Option	Description
Yes	Saves the drawing. Displays the Save Drawing As dialog box; enter a file name, and then click Save. See **SaveAs** command.
No	Does not save the drawing. Closes the drawing without saving.
Cancel	Cancels the command. Returns to the drawing; does not save or close it.

Related Commands

WCascade - cascades the windows.
Close - closes the current drawing.
WCloseAll - closes all windows.
WhTile - tiles windows horizontally.
WiArrange - arranges iconized windows.
WvTile - tiles windows vertically.
SysWindows - arranges windows.

WCloseAll

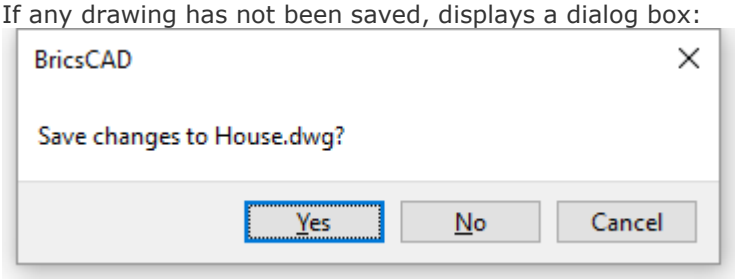
Closes all windows (short for "windows close all").

Accessing the Command

command bar: *wcloseall*
alias: *closeall*
menu bar: *Window | Close All*

: ***wcloseall***

If any drawing has not been saved, displays a dialog box:



Click a button.

This dialog box repeats for every unsaved drawing.

Command Options

Option	Description
Yes	Saves the drawing. Displays the Save Drawing As dialog box; enter a file name, and then click **Save**. See **SaveAs** command.
No	Does not save the drawing. Closes the drawing without saving.
Cancel	Cancels the command. Returns to the drawing; does not save or close it.

Related Commands

WCascade - cascades the windows.
WClose - closes the current windows.
WhTile - tiles windows horizontally.
WiArrange - arranges iconized windows.
WvTile - tiles windows vertically.
SysWindows - arranges windows.

Weblight

Creates a web light: 3D representation of the light intensity distribution of a light source.

Accessing the Command

command bar: *weblight*
menu bar: *View | Rendering | Lights | New web light*
toolbar: *Lights |*

: ***weblight***
Prompts you in the command bar:

If the LIGHTINGUNITS system variable is 0 (No lighting units), the following prompt displays:

Command not allowed when LIGHTINGUNITS = 0

If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), the following prompt displays:

Specify source position <0,0,0>: specify the position of the light.)

Specify target position <0,0,-10>: (specify the target point of the light.)

Enter an option to change [Name/Intensity factor/Status/Photometry/weB/shadoW/filterColor/eXit] <eXit>:

Command Options

Option	Description
Name	Enter light name <Spotlight1>: (Enter a name or press Enter to accept the default name.)
Intensity Factor	Enter intensity (0.00 - max float) <1.0>: (Enter the intensity. Values between 0.00 and the maximum value supported by your system are accepted.)
Status	Enter status OFf/<ON>: (Choose ON to include the light when calculating a rendering.)

Photometry	Enter a photometric option to change: Color/eXit/<Intensity>:
	• Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>: (Enter the intensity expressed in Candela (Cd).)
	  • Enter Flux (Lm) : (Enter the luminous flux expressed in Lumen (Lm).)
	  • Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>: (Enter the illuminance expressed in Lux (Lx) or Foot-candle (Fc).)
	  • Enter Distance <1.0>: (Enter a distance expressed in drawing units.)
	• Enter color name or enter an option [?/Kelvin] <D65>:
	  • Enter color name(s) to list <*>: (Enter a color name from the list. Type * to see the list)
	  • D65
	  • Fluorescent
	  • Coolwhite
	  • Whitefluorescent
	  • daylightfluorescenT

- Incandescent
- Xenon
- Halogen
- Quartz
- Metalhalide
- mErcury
- Phosphormercury
- highpressureSodium
- Lowpressuresodium

- Enter Kelvin temperature <3600.0>: (Enter a color temperature expressed in degrees Kelvin (K).

Candela: The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela.

Lumen: The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr

As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens.

Illuminance: In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human brightness perception.

If LIGHTINGUNITS=2 (International lighting units):
In SI derived units illuminance is measured in lux (lx) or lumens per square metre.

If LIGHTINGUNITS=1 (American lighting units):
The foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux, although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux.

Kelvin temperature: Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K).

	Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700&endash;3,000 K) are called warm colors (yellowish white through red).
Web	Enter a Web options to change: File/X/Y/Z/<Exit>: (Enter an option.) • Enter Web file <>: Enter the name of a web file. An IES file is a text file that describes the intensity of a light source at points on a spherical grid. • Enter Web X rotation in degrees <0.0000>: (Define the X rotation for the web.) • Enter Web Y rotation in degrees <0.0000>: (Define the Y rotation for the web.) • Enter Web Z rotation in degrees <0.0000>: (Define the Z rotation for the web.)

Shadow	Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>: (Enter a shadow setting.)
	• Off: Disables the calculation of shadows for the light.
	• Sharp: Displays shadows with sharp edges. Use this option to increase performance.
	• Soft mapped: Displays realistic shadows with soft edges.
	Prompts you: Enter map size [64/128/256/512/1024/2048/4096] <256>:
	• Soft sampled: Displays realistic shadows with softer shadows based on extended light sources.
	Prompts you: Enter an option to change Shape/sAmples/Visible/<eXit>:
	  • Enter shape [Disk/Rect] <Disk>:
	  • Enter radius <0.0>: (Enter the radius of the disk.)
	  • Enter Length <0.0>: (Enter length of the rectangle.)
	  • Enter Width <0.0>: (Enter width of the rectangle.)
	  • Enter Shadow Sample <16>:
	  • Enter Shape Visibility [Yes/No] <No>:

Filter Color	Enter true color (R,G,B) or enter an option: Index color/Hsl/<255,255,255>: Defines the color of the light. • True Color (RGB): The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • Index color: Specifies an AutoCAD Color Index color. • HSL: Specifies the color using three parameters Hue, Saturation and Lightness.
Exit	Concludes the command and creates the light.

Related Commands

Distantlight - creates a distant light.
Light - creates a new light in the drawing.
Lightlist - displays the Drawing Explorer - Lights dialog window.
Pointlight - creates a point light.
Render - calculates a photorealistic image of the 3D model.
Spotlight - creates a spot light.
Sunproperties - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.

Wedge

Creates a wedge as a polyface mesh (Classic); creates a wedge as a 3D solid (Pro and Platinum).

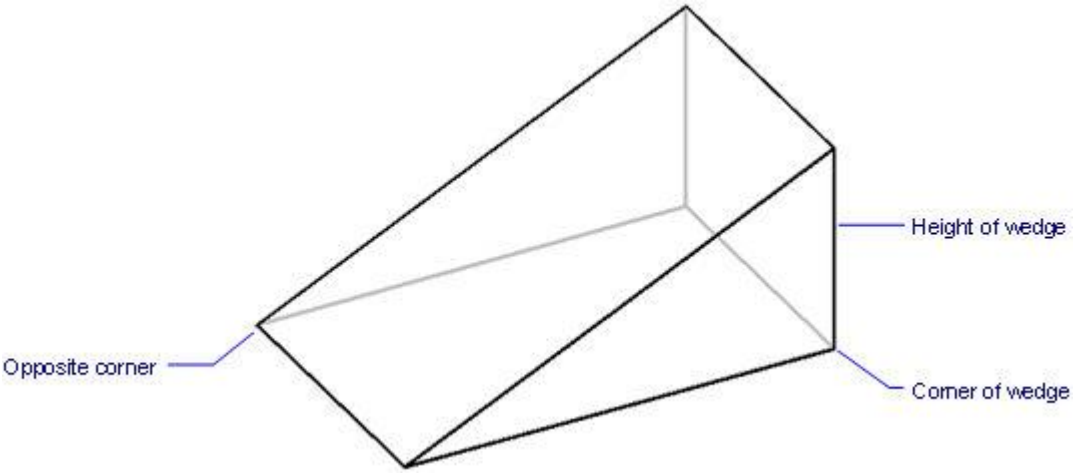
Accessing the Command

command bar: *wedge*
alias: *we*
menu bar: *Model| 3D Soldis | Wedge*
toolbar: *3D Solids |*

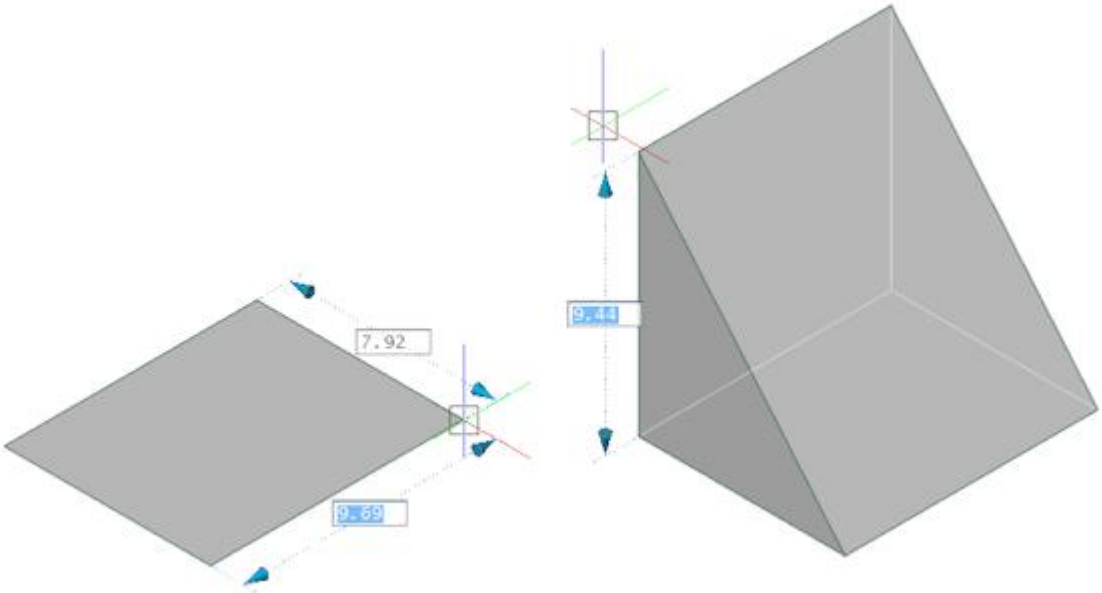
: ***wedge***
Prompts you in the command bar:

Set corner of wedge or [Center] <4.25,0.00,1.84>: (Pick a point, or enter coordinates.)

Set opposite corner or [Cube/Length of side]: (Pick another point, or enter an option.)
Height of wedge **[**2Point**]:** (Specify the height, or enter an option.)

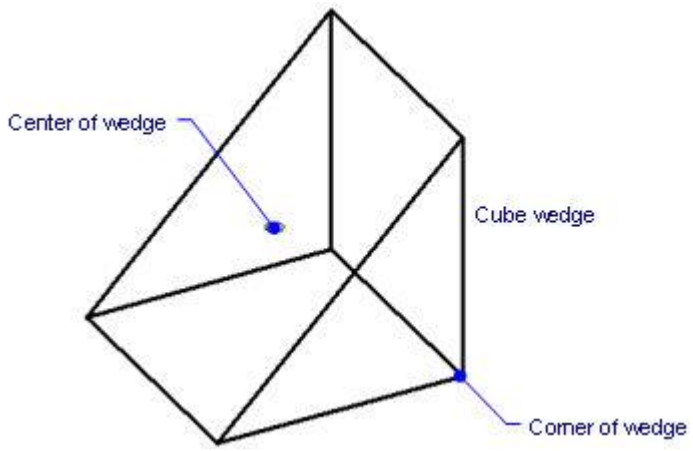
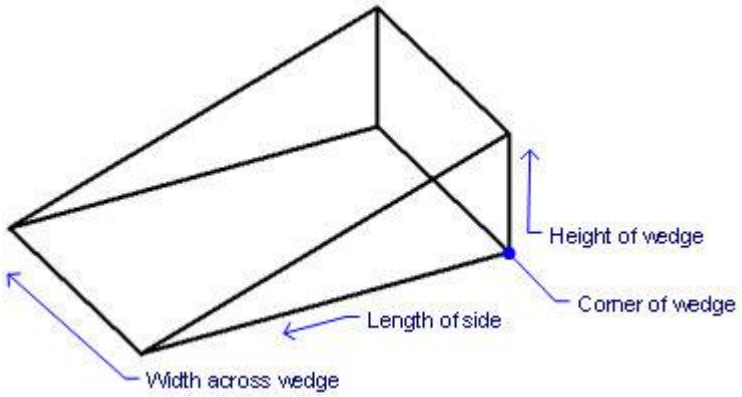


If Dynamic Dimensions are active, you can enter the dimensions in the dynamic dimensions entry fields.



Command Options

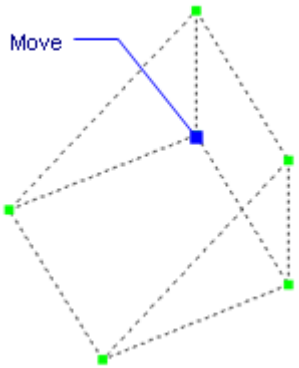
Option	Description
Corner of wedge	Specifies one corner of the wedge's base.
Opposite corner	Specifies the other corner of the base.
Height of wedge	Specifies the height of the wedge. Note that the tall end of the wedge is drawn from the first corner point specified. Choose the 2Point option to specify the height of the wedge by picking two points.
Center	Draws the wedge using a center point and a corner. Prompts you: **Center of wedge** - specify the center of the entire wedge (not the base). **Corner of wedge** - specify the location of one of the wedge's corners.
Cube	Draws a wedge with equal length, width, and height. Prompts you: **Length of sides of cubic wedge** - specify the same distance for each dimension, length, width, and height.

	
Length 	Allows you to enter distances of all three dimensions; don't enter coordinates. Prompts you: **Length of side** - specify the length in the x direction; enter a negative number to draw the wedge in the negative x direction. **Width across wedge** - specify the width of the base. **Height of wedge** - specify the height.

Grips Editing

Wedges can be edited directly through grips:

1. Select the wedge. Notice that it has several grips.
2. Drag a grip to move the wedge. (All grips perform the same function: move.)



Related Commands

Box - creates 3D boxes as solid models.
Cone - creates 3D cones as solid models.
Cylinder - creates 3D cylinders as solid models.
Dish - creates the lower half of 3D balls as solid models.
Dome - creates the upper half of 3D balls as solid models.
Extrude - extrudes 2D entities into 3D solid models.
Interfere - creates 3D volumes of intersecting 3D solid models.
Intersect - removes volumes not in common between two solid models.
MassProp - reports the mass properties of 3D solid models and 2D regions.
Pyramid - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic)
Revolve - revolves 2D entities into 3D solid models.
Slice - cuts 3D solids into two parts.
SolidEdit - edits the bodies, faces, and edges of 3D models.
Sphere - creates 3D balls as solid models.
Subtract - removes one solid model from a second.
Torus - creates 3D donuts as solid models.
Union - joins two or more solid models into a single entity.
Wedge - creates 3D wedges as solid models.

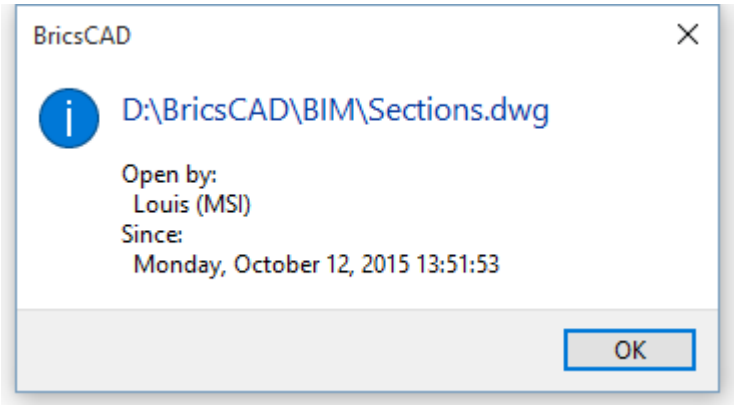
WhoHas

Reports the ownership of a drawing file.

Accessing the Command

command bar: *whohas*

: ***whohas***
Displays a message box:



The information in the message box is stored in a temporary DWL (Drawing Lock) File, which is deleted when the drawing is closed.

Command Options

No Options.

Related Commands

DwgProps - opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.
Open - opens drawing (DWG), template (DWT), and interchange (DXF) files.
SecurityOptions - sets a password to protect a drawing

WhTile

Tiles windows horizontally (short for "window horizontal tile").

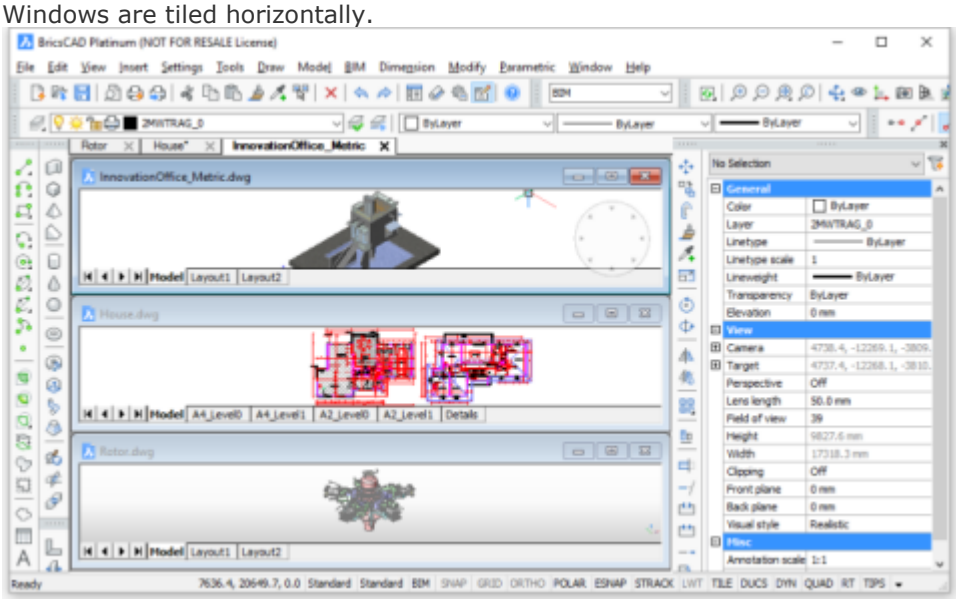
Accessing the Command

command bar: *whtile*
menu bar: *Window | Tile Horizontally*

: ***whtile***

Displays no prompt in the command bar.

Windows are tiled horizontally.



Command Options

No options.

Related Commands

WCascade - cascades the windows.
WClose - closes the current window.
WCloseAll - closes all windows.
WiArrange - arranges iconized windows.
WvTile - tiles windows vertically.
SysWindows - arranges windows.

WiArrange

Lines up iconized windows (short for "window icon arrange").

Accessing the Command

command bar: *wiarrange*

: ***wiarrange***

Displays no prompt in the command bar.

Arranges the icons of minimized windows at the bottom of the graphical screen.

This command works only when windows are minimized as icons.

Command Options

No options.

Related Commands

WCascade - cascades the windows.
WClose - closes the current window.
WCloseAll - closes all windows.
WhTile - tiles windows horizontally.
WvTile - tiles windows vertically.
SysWindows - arranges windows.

WipeOut

Creates wipeouts to mask part of a drawing and controls the display of wipeout frames in the drawing.

Accessing the Command

command bar: *wipeout*
menu bar: *Draw | WipeOut*
toolbar: *Draw |*

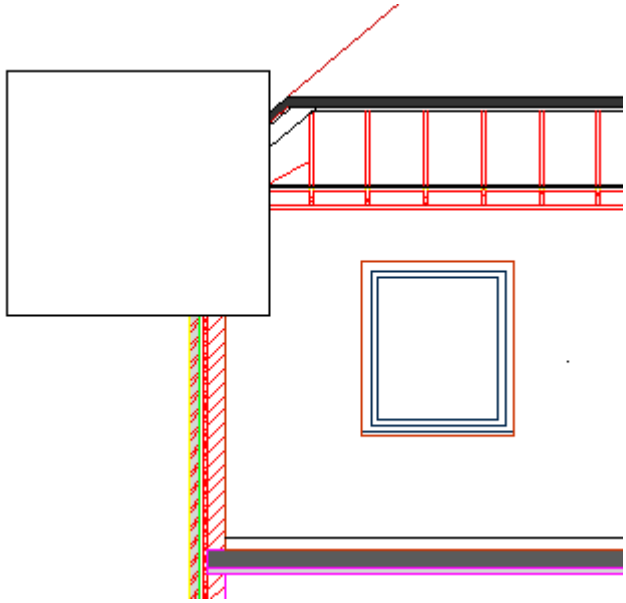
: ***wipeout***
Prompts you in the command bar:

Specify start point or [Polyline/Frames]: (Pick a point, or enter an option.)

Specify next point: (Pick another point.)

Specify next point or [Undo]: (Pick another point, or type U.)

Specify next point or [Undo/Close]: (Enter another point, or type C to end the command.)



Use the DrawOrder command to move the wipeout in front or behind other entities.

Command Options

Option	Description
Undo	Removes the last segment.
Close	Creates the wipeout.
Frames	Toggles the display of all wipeout frames in the current drawing; prompts you: Enter frame setting [ON/OFf/Display but not plot] <ON> - choose an option. • OFF - turns off frames. When the fame is off, the wipeout cannot be selected. • ON - turns on frames. Wipeouts can be selected and are printed. • Display but not plot - turns on frames. Wipeouts can be selected but are not printed.
Polyline	Uses a closed polyline to construct the wipeout frame; prompts you: **Select a closed polyline** - pick a closed polyline. Only polylines composed of line segments are accepted. Delete selected polyline? [Yes/No] <No>: - determines whether the polyline is erased: • Yes - erases the polyline. • No **- does not erase the polyline.**

Procedures

To create a wipeout using points
To create a wipeout from a polyline
To control the display of the edges of wipeouts

Grips Editing

Wipeouts can be edited directly through grips:

1. Select the wipeout. Notice that it has one grip at each vertex.
2. Drag a grip to stretch the vertex.

Related Commands

DrawOrder - changes the display order of overlapping entities.
PLine - draws polylines.
RevCloud - draws revision clouds.

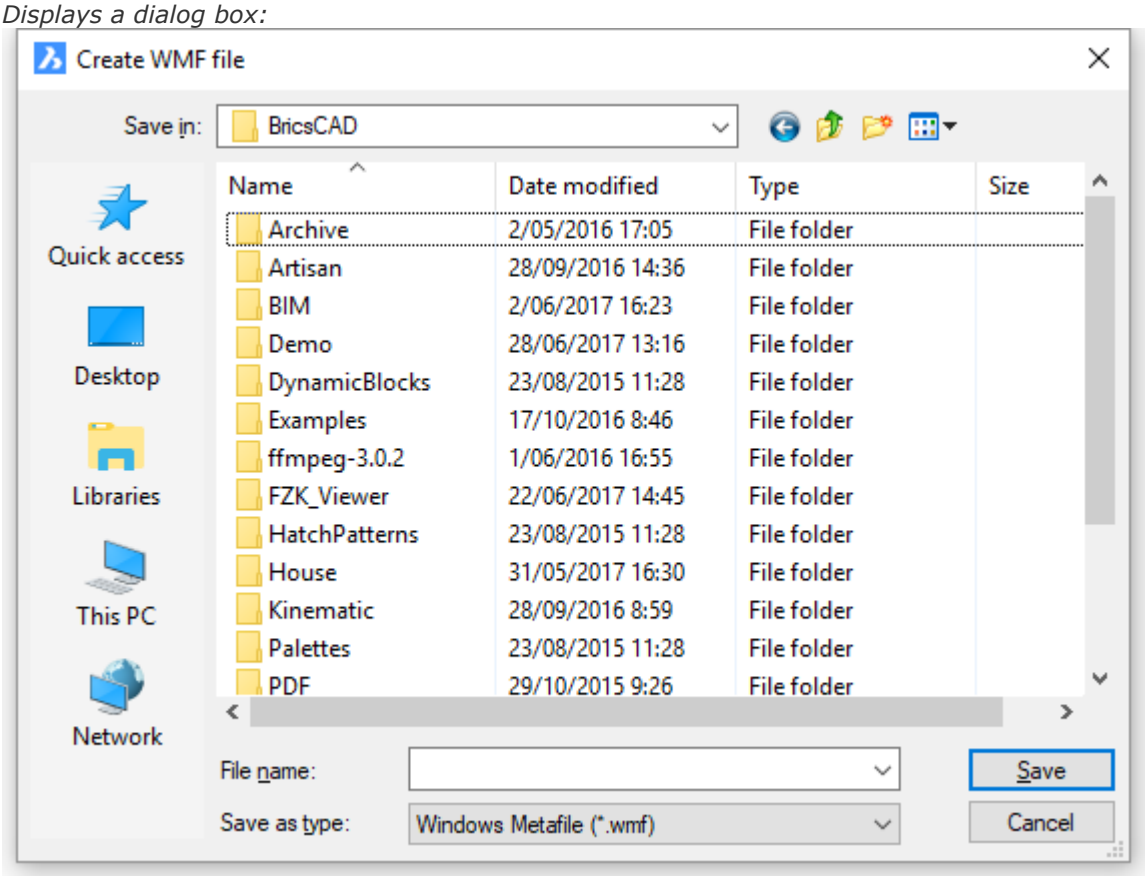
WmfOut

Exports drawings in WMF, EMF, or SLD format (short for "windows metafile output").
On the Linux platform use MSlide to create slide (SLS) files.

Accessing the Command

command bar: *wmfout*
alias: *wo*

: ***wmfout***
Displays a dialog box:



Enter a file name, choose a file type, and then click Save.

WMF, EMF, and SLD files can be viewed with the **VSlide** command.

Command Options

Option	Description
File name	Specifies the name of the slide (snapshot).
Save as type	Specifies the file format: • SLD - compatible with AutoCAD. • EMF - enhanced WMF format. • WMF - Windows meta format, a hybrid of vector and raster formats. The *PictureExportScale* user preference sets the view size when exporting to WMF or EMF.
Save	Saves the file.

Related Commands

BmpOut - exports drawings in BMP format.
Export - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats.
MSlide - exports drawings in WMF, EMF, and SLD formats.

Wnext

Displays the next drawing window if multiple drawings are open simultaneously. Short for 'window next.

Accessing the Command

command bar: *wnext*
keyboard: Ctrl + Tab

: ***wnext***

Command Options

No command options.

Related Commands

Wprev - Displays the previous drawing window if multiple drawings are open simultaneously.

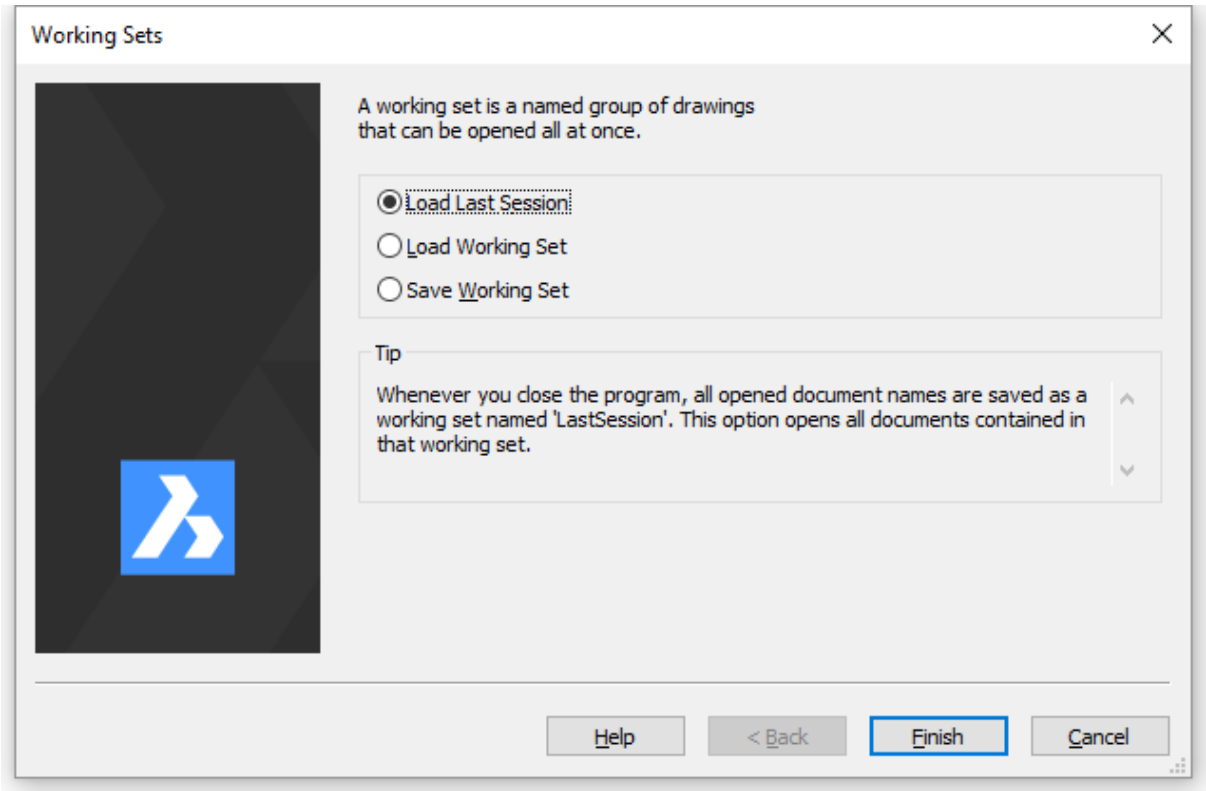
WorkSets

Creates and loads a set of drawing files. Allows to work with a named group of drawings.

Accessing the Command

command bar: *worksets*
menu bar: *File | Working Sets...*

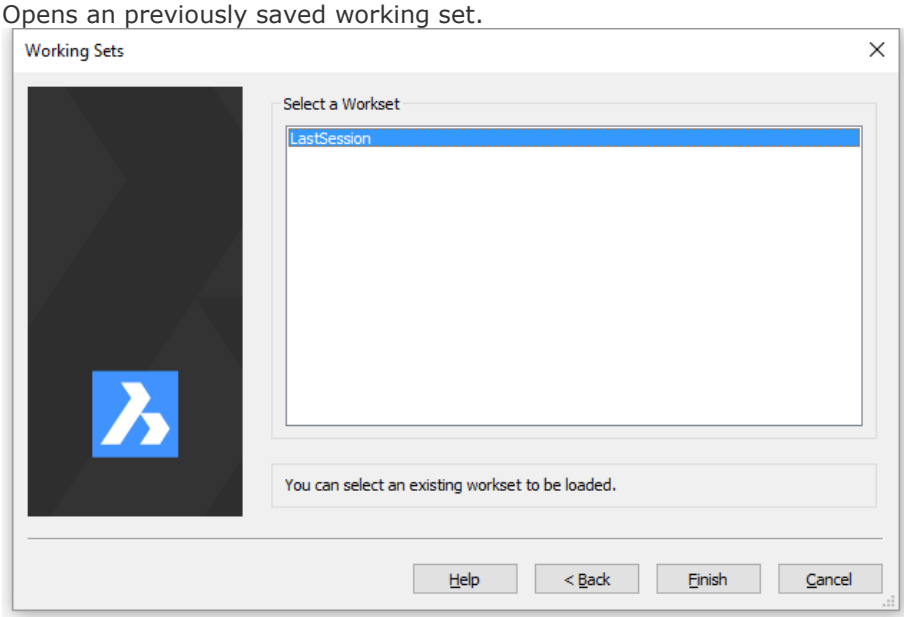
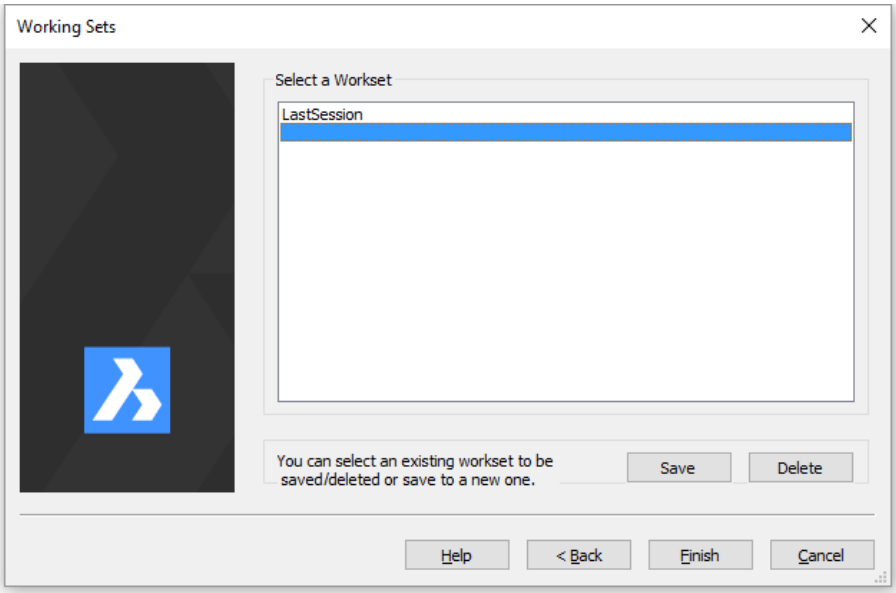
: ***worksets***
Displays a dialog box:



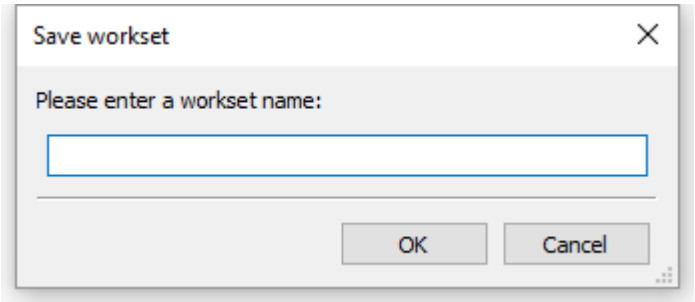
Select an option, then click the *Next* or *Finish* button.

NOTE Since working sets are saved in the registry, they are defined differently in each user profile.
See the ProfileManager command to learn more about user profiles.

Command Options

Option	Description
Load Last Session	Opens all drawings that where open when the last BricsCAD session was closed. Click the *Finish* button to confirm.
Load Working Set	Opens an previously saved working set.  Select a named working set in the list, then press the *Finish* button.
Save Working Set	Allows to: • Save the currently loaded drawings as a named working set. • Redefine (overwrite) an existing named working set. • Delete an existing named working set.  Do one of the following:

- Click the *Save* button.
  Displays a dialog box:



  Type a name in the *Workset Name* field, then press the *OK* button.

  Click the *Finish* button to conclude.

- Select an existing named working set, then click the *Save* button.
  A *Confirm Save* dialog box displays.
  Click the *Yes* button to replace the selected set with the currently open drawings.

  Click the *Finish* button to conclude.

- Select an existing named working set, then click the *Delete* button.
  A *Confirm Delete* dialog box displays.
  Click the *Yes* button to delete the selected set.

  Click the *Finish* button to conclude.

Related Commands

Open - opens drawing (DWG), template (DWT), and interchange (DXF) files.

Workspace

Sets the current workspace and lets you create, modify and save workspaces.

Accessing the Command

command bar: *workspace*

: ***workspace***
Prompts you in the command bar:

Workspace [set Current/Save As/Rename/Delete/SEttings/?] <Set Current>: (Select an option)

Command Options

Option	Description
Set Current	Sets the current workspace. Prompts you: Enter workspace name to make current [?] <CURRENT_WORKSPACE>: Do one of the following: • Type *?* to list the existing workspaces in the command window. • Type the name of the workspace in the command window.
Save As	Saves the current settings to a new workspace. Prompts you: Save current workspace as <CURRENT_WORKSPACE>: Do one of the following: • Press Enter to save the current workspace. • Type a name, then press Enter, to create a new workspace.
Rename	Renames a workspace. Prompts you: Enter workspace to rename [?] <CURRENT_WORKSPACE>: Do one of the following: • Type *?* to list the existing workspaces in the command window. • Press Enter to rename the current workspace. Enter new workspace name <CURRENT_WORKSPACE>: Type a new name in the command window, then press Enter. • Type the name of an existing workspace, then press Enter. Enter new workspace name <WORKSPACE>: Type a new name in the command window, then press Enter.
Delete	Deletes a workspace. Prompts you: Enter workspace to delete [?]: Do one of the following: • Type *?* to list the existing workspaces in the command window. • Type the name of an existing workspace, then press Enter. Do you really want to delete the workspace "WORKSPACE"? [Yes/No] <N>: Type *Y*, then press Enter to delete the workspace.
Settings	Opens the *Customize > Workspaces* dialog. See Managing Workspaces in the *User Guide*.
?	Lists the existing workspaces in the command window.

Procedures

Managing workspaces
Setting the current workspace
Creating a workspace
Editing a workspace
Deleting a workspace

Related Commands

Customize - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, mouse actions, tablet, Quad settings, workspaces, aliases, and shell commands.
WsSave - saves the current workspace under a different name, thus creating a new workspace.

Wprev

Displays the previous drawing window if multiple drawings are open simultaneously. Short for 'window previous'.

Accessing the Command

command bar: *wprev*
keyboard: Ctrl + Shift + Tab

: ***wnext***

Command Options

No command options.

Related Commands

Wnext - displays the next drawing window if multiple drawings are open simultaneously.

WsSave

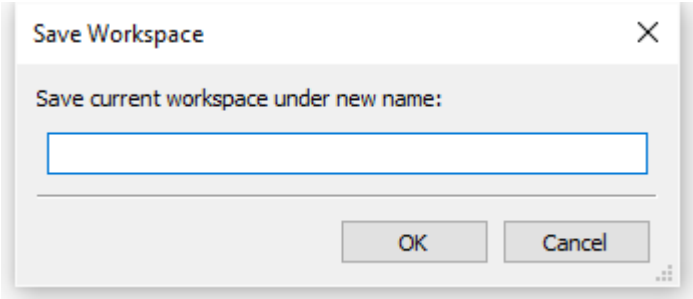
Saves the current workspace under a different name, thus creating a new workspace.

Accessing the Command

command bar: *wssave*
menu: *Tools | Save Workspace...*

: ***wssave***

Displays a dialog box:



Type a name, then press the OK button.

Command Options

No command options.

Related Commands

Customize - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, mouse actions, tablet, Quad settings, workspaces, aliases, and shell commands.
Workspace - sets the current workspace and lets you create, modify and save workspaces.

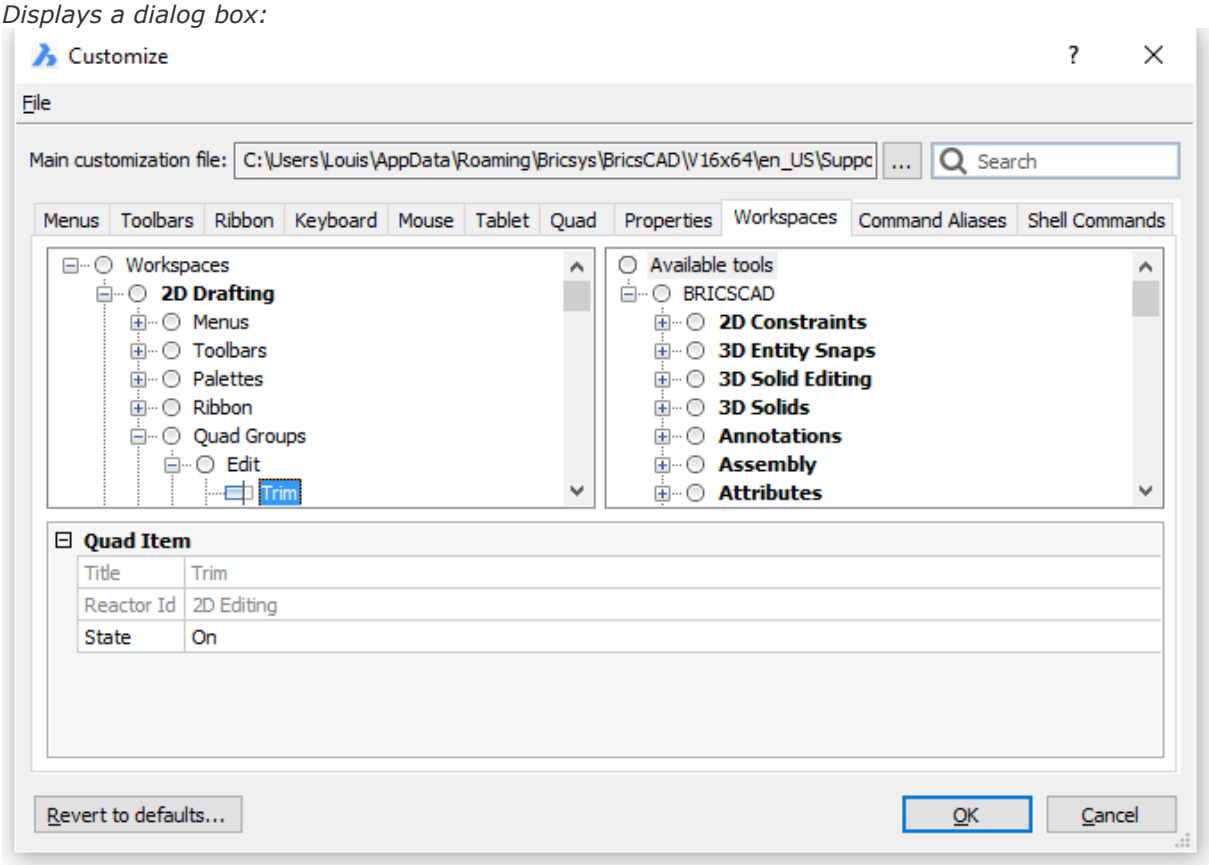
WsSettings

Opens the Customize Workspaces dialog box.

Accessing the Command

command bar: *wssettings*

: ***wssettings***
Displays a dialog box:



Command Options

See Customize Workspaces.

Related Commands

Customize - customizes the user interface of BricsCAD: Menus, Toolbars, Keyboard shortcuts, Mouse actions, tablet, Quad settings, Properties, Workspace settings, Aliases, and Shell Commands.

WvTile

Tiles windows vertically (short for "window vertical tile").

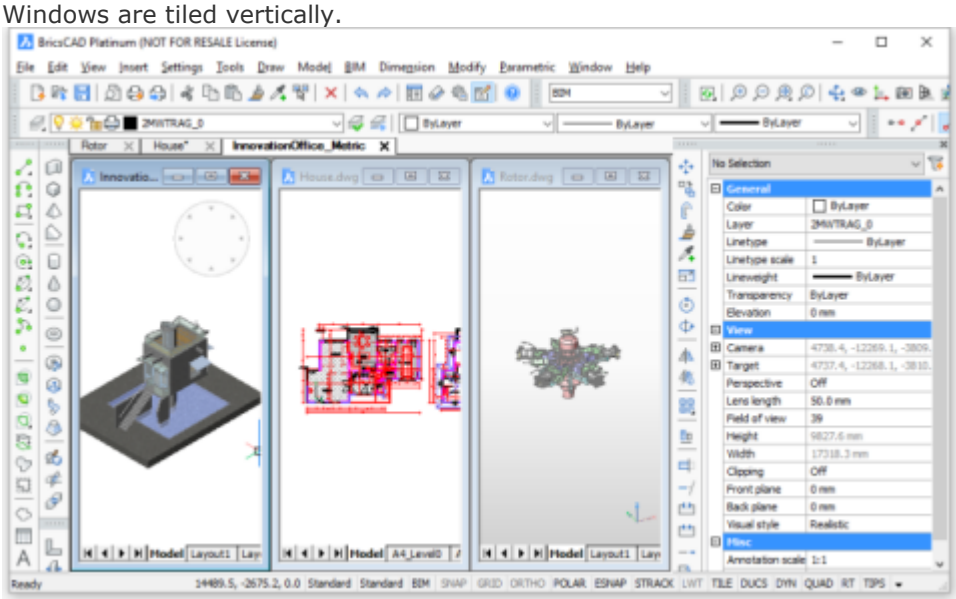
Accessing the Command

command bar: *wvtile*
menu bar: *Window | Tile Vertically*

: ***wvtile***

Displays no prompt in the command bar.

Windows are tiled vertically.



Command Options

No options.

Related Commands

WCascade - cascades the windows.
WClose - closes the current window.
WCloseAll - closes all windows.
WhTile - tiles windows horizontally.
WiArrange - arranges iconized windows.
SysWindows - arranges windows.

X

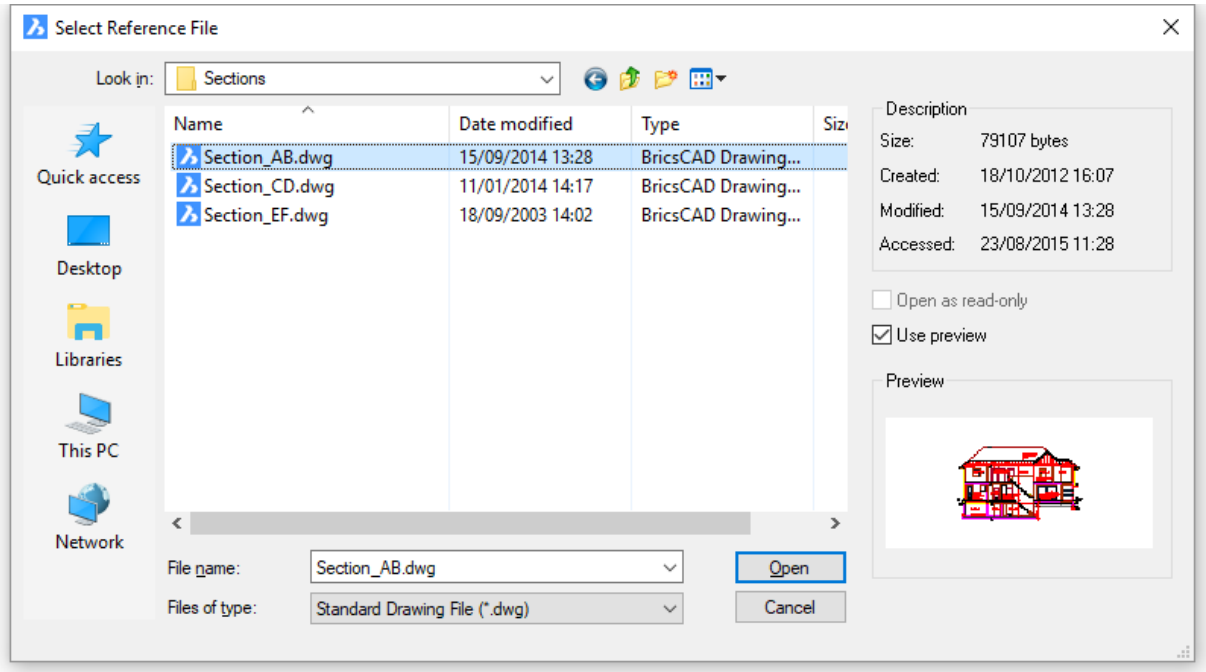
XAttach

Attaches externally-referenced drawings through a dialog box (short for "external attach").

Accessing the Command

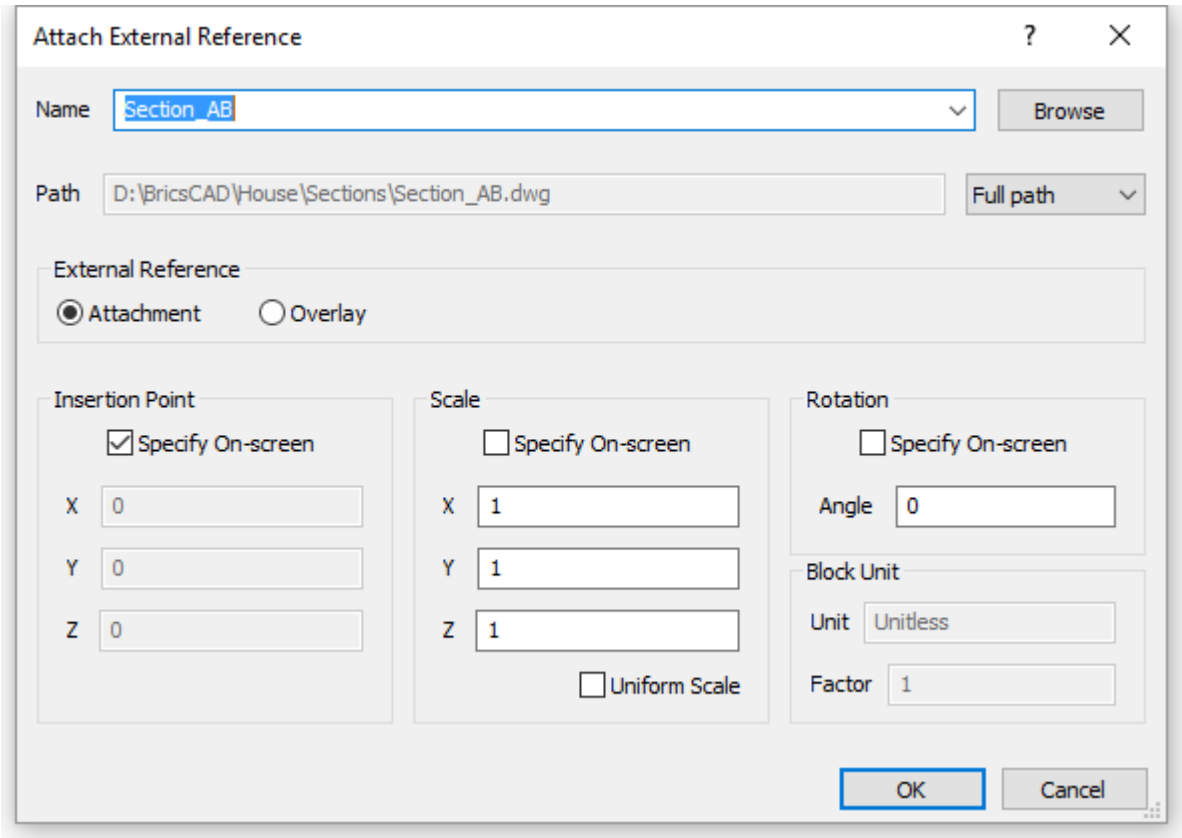
command bar: *xattach*
alias: *xa*
menu bar: *Insert | Attach XRef*
toolbar: *Insert |*

: ***xattach***
Displays a dialog box:



Choose a DWG file, and then click Open.

BricsCAD displays a dialog box:



Choose options, and then click OK.

If you choose Specify On-screen, then you will be prompted in the command bar for the insertion point, scale, and/or rotation angle.

The XDWGFADECTL system variable controls the fading level for Xref inserts. Non positive values disable fading. Values between -90 and 90 are accepted.

This command is a shortcut to using the -XRef command's Attach option.

Command Options

Option	Description
Name	Specifies the name of the attached drawing file. If more than one DWG file has been attached to the drawing, their names are listed by the droplist: Choosing another name lets you modify the xref's properties.
Browse	Displays the Open dialog box. Choose a DWG file, and then click Open. Choosing another file lets you attach a additional drawings without leaving this dialog box.

Path	Reports the path that will be remembered by BricsCAD. You cannot edit this field; instead, use the Browse button to modify the path to the file.
Full Path	Click the droplist to choose a different path type: Full path - stores the drive and folder(s) names to the DWG file (absolute reference), such as: c:\cad\dwg\filename.dwg Relative path - stores the route to the xref'ed DWF file's folder, such as: ..\filename.dwg (..\ means one folder higher.) This option does not work until the drawing has been saved at least once. No path - strips the drive and folder(s) names, leaving just the DWG file name, such as: filename.dwg
External Reference	Specifies how the external reference is to be attached: Attachment - attaches the xref and nested xrefs. Overlay - attaches only the first level of xref.
Insertion Point	Specifies the location of the xref's lower-left corner.
Specify On-screen	Determines when the insertion point is specified: On - specify the insertion point in the drawing after the dialog box is closed; you will be prompted: **Insertion point for block** - pick a point or enter x,y,z coordinates. Off - specify the insertion point in the dialog box using the X, Y, and Z fields.
X Y Z	Specifies the x,y,z coordinates for the xref's insertion point. Use 0,0,0 to insert the xref at the drawing's origin.
Scale	Specifies the size of the xref.
Specify On-screen	Determines when the scale factors are specified: On - specify the scale factor in the drawing after the dialog box is closed; you will be prompted: Corner/XYZ/X scale factor - enter an option, pick two points a number, or press Enter for the default value. Y scale factor: < Equal to X scale> - enter an option, a number, or press Enter for the default value Off - specify the scale factor in the dialog box using the X, Y, and Z fields.
X Y Z	Specifies the 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 that of x.

Rotation	Specifies the rotation angle of the xref.
Specify On-screen	Determines when the rotation angle is specified: On - specify the angle in the drawing after the dialog box is closed; you will be prompted: Rotation angle for block - enter a number, pick two points, or press Enter for the default value. Off - specify the angle in the dialog box using the Angle field.
Angle	Specifies the rotation angle about the insertion point. Positive angles rotate the xref counterclockwise. Use 0 to keep the xref at it original orientation.
Block Unit	Unit: shows the INSUNITS setting of the drawing being attached. Factor: indicates the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.

Procedures

To attach an external reference

Related Commands

Insert - inserts drawings as blocks.
PlaceView - places a model view selected from a source drawing into a paper space layout of the current drawing.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
-XRef - attaches and controls xrefs through the command bar.
XClip - clips externally-referenced drawings.

XClip

Clips externally-referenced drawings, and adjust front and rear clipping planes (short for "xref clip").

Accessing the Command

command bar: *xclip*
alias: *clip*
menu bar: *Modify | XRef Clip*

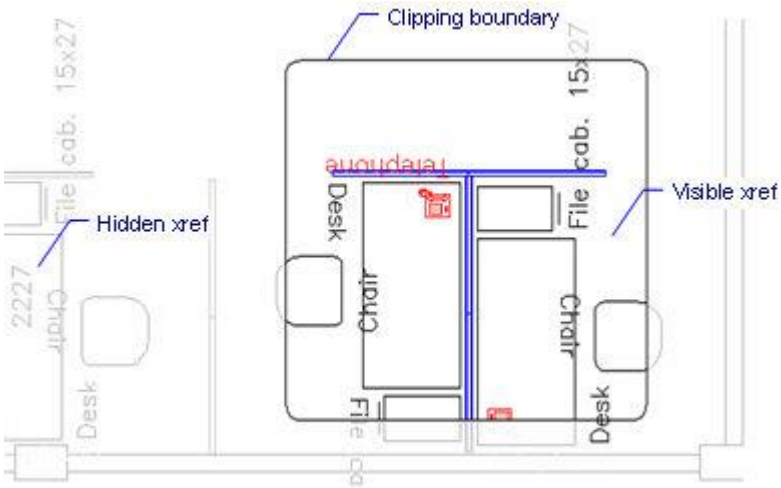
: ***xclip***
Prompts you in the command bar:

Select entities: (Choose one or more xrefs.)
Entities in set: 1

Select entities: (Choose more xrefs, or press Enter to continue.)

[ON/OFf/Clipdepth/Invert/Delete/generate Polyline/New boundary] <New boundary>: (Enter an option, or press Enter to create a new clipping boundary.)

[Select polyline/Polygonal/Rectangular] <Rectangular>: (Enter an option or press Enter to create a rectangular clipping boundary.)



The hidden portion of the xref is shown in gray for illustrative purposes only; it is normally not seen when clipping is turned on.

Command Options

Option	Description
New boundary	depending on whether or not a boundary already exits the command will produce following prompts: If no boundary exists you are prompted: [Select polyline/Polygonal/Rectangular] - type S, P, or R. If a boundary already exists you are prompted: Delete old boundary(s)? [Yes/No] - type Y or N. This option replaces existing clipping boundaries with a new one. Please note that the command

	will only continue (by prompting you to select a new boundary) after all previous boundaries are deleted. So typing No at previous prompt will cancel the Xclip command.
Select polyline	Choose an existing polyline. You can pick splined polylines, rectangles, donuts, and polygons. The polyline does not need to be closed, but cannot cross over itself.
Polygonal	Creates multi-sided clipping boundaries; prompts you: Specify first point - pick a point in the drawing, or specify x,y coordinates. Specify next point or [Undo] - pick another point in the drawing, or specify x,y coordinates. Specify next point or [Undo] - pick more points, or press Enter to complete the polygon.
Rectangular	Creates rectangular clipping boundaries; prompts you: **Specify first corner** - pick a point in the drawing, or specify x,y coordinates. **Specify opposite corner** - pick another point in the drawing, or specify x,y coordinates.
ON	Turns on clipping boundaries.
OFF	Turns off clipping boundaries; all of the xref is again visible. The clipping boundary is saved though.
Clipdepth	Sets 3D clipping, front and rear; prompts you: Specify front clip point or [Distance/Remove] - enter an option. Specify back clip point or [Distance/Remove] - enter an option. This option is used for 3D xrefs, where some of the front and back of the model is to be removed from view. This option does not operate until you create at least one clipping boundary.
Specify front clip point	Specifies the position of the front clipping plane. Pick a point, or enter coordinates. The plane is parallel to the current viewpoint.
Distance	Specifies the distance of the clipping point; prompts you: Specify distance from boundary - specify the distance from the clipping boundary to the front or rear plane.\
Remove	Removes the front or rear clipping point.
Specify back clip point	Specifies the position of the back clipping plane. Pick a point, or enter coordinates. The plane is parallel to the current viewpoint.

Invert	Inverts the clipping mode: selected Xrefs are clipped either outside or inside the clipping boundary.
Delete	Erases all clipping boundaries; does not prompt or warn you.
Generate Polyline	Traces a polyline over a selected clipping boundary. This allows you to edit the clipping boundary, but in an indirect fashion: 1. Generate the polyline with this option, and then exit the command. 2. Use the **PEdit** command to edit the polyline. 3. Repeat the XClip command, and then use the New | Select polyline option to turn the polyline back into a clipping boundary.

Related Commands

pdfclip - clips a PDF underlay.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
-XRef- attaches and controls externally referenced drawings through the command bar.
VpClip - clips viewports.
ImageClip - clips images.

XEdges

Creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, and regions.

Accessing the Command

command bar: *xedges*
menu bar: *Draw | Boundaries | Extract Edges*
toolbar: *Draw | Boundaries |*
Quad: *Draw |*

: ***xedges***
Prompts you in the command bar:

Select entities: (Select a 3D solid, 3D solid face or a region.)
Entities in set: 1

Next Prompt: (Select more entities or right click.)

The 2D linear entities generated from the edges of the selected 3D entities are placed on the current layer, using the current entity creation settings.

Command Options

No command options.

Related Commands

Boundary - creates a polyline that forms a boundary inside a closed area through a dialog box.
-Boundary - creates a polyline boundary inside closed areas through the command bar.
Imprint: imprints 2D entities onto planar faces of 3D solids and surfaces; allows to create additional edges on planar faces.
ProjectGeometry - projects 2D geometry onto a region, a surface or a 3D solid; allows to create additional linework on the faces of solids and surfaces.
XFaces - extracts a set of 3D solid or surface faces as a new 3D surface object.

XFaces

Extracts a set of 3D solid or surface faces as a new 3D surface object.

Accessing the Command

command bar: *xfaces*
Quad: *Draw* |

: ***xfaces***
Prompts you in the command bar:
Select faces to extract [Move]: (Select faces*.)
Select faces to extract [Move]: (Select more faces or press Enter to extract the current selection set.)

*You can use the window selection methods to select the faces. The *Select Faces* mode is used automatically as indicated by the icon next to the cursor.

The newly created 3D surfaces are placed on the current layer, using the current entity creation settings.

Command Options

Option	Description
Move	Allows to move the newly created 3D surface object(s). Prompts you: Enter base point [Displacement] <Displacement>: Specify a point. Enter second point <Use base point as displacement>: Specify a point.

Related Commands

XEdges - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, and regions.

XLine

Draws infinitely long lines.

Accessing the Command

command bar: *xline*
aliases: *xl, infline*
menu bar: *Draw | Infinite Line*
toolbar: *Draw | Lines |*

: *xline*
Prompts you in the command bar:

Infinite line: Set point along line or [Horizontal/Vertical/Angle/Bisect/Parallel]: (Pick a point or enter an option.)

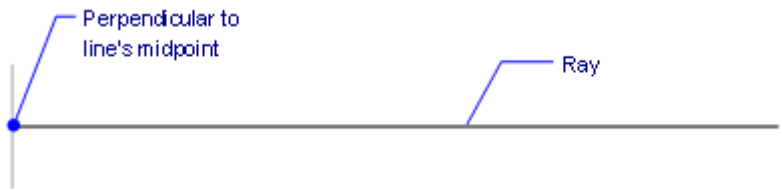
Direction: (Pick a point, or enter x,y coordinates.)

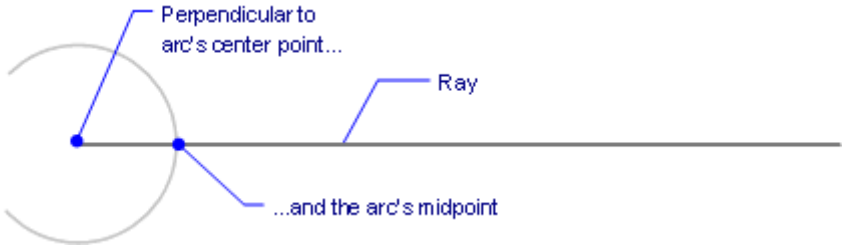
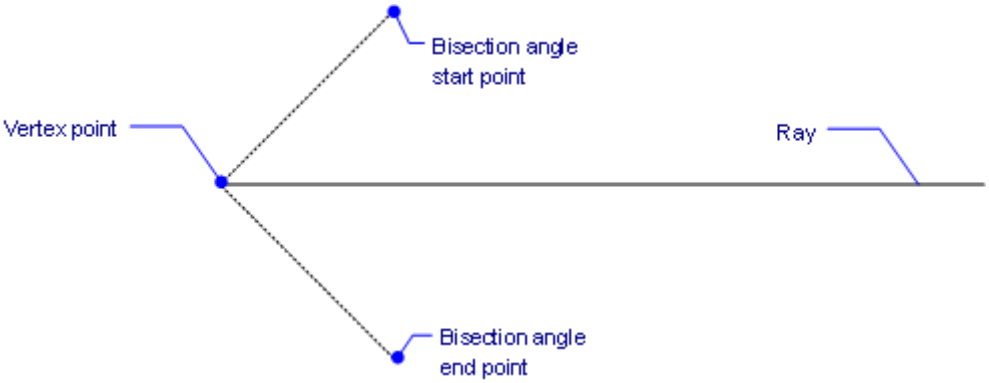
Direction: (Pick another point, or press Enter to exit command.)

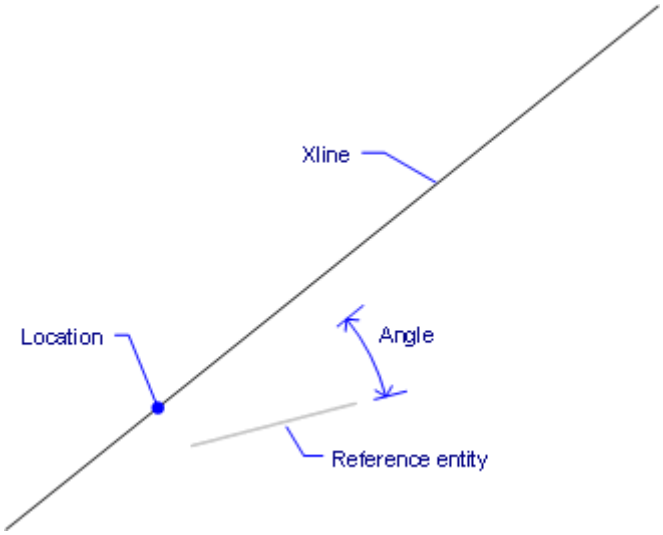
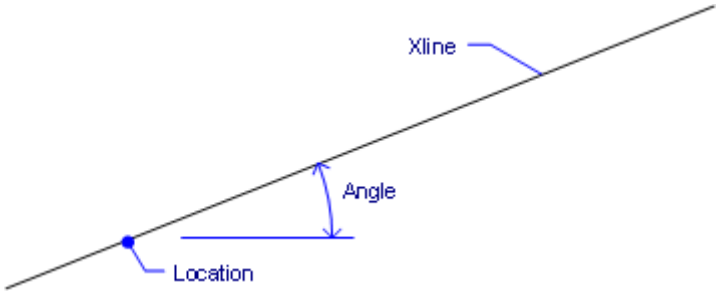
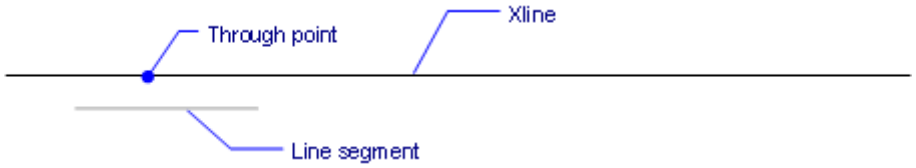
An xline has no starting point and no ending point; both ends stretch to infinity.

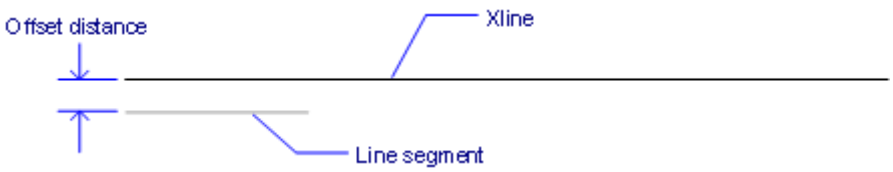
Xlines are ignored when BricsCAD determines the extents of the drawing.

Command Options

Option	Description
Point along line	Specifies a point on the xline. Pick a point in the drawing, or enter x,y coordinates; prompts you: Direction - specify the direction of the xline. The Direction prompt repeats so that you can draw multiple xline from the same starting point. Press Enter to exit the command.
Bisect	Draws an xline that bisects entities perpendicularly; prompts you: Entity/<Vertex point> - enter an option.
Entity	Draws xlines perpendicular to the midpoint of lines and polyline segments, and the center point of arcs and polyarcs; prompts you: Select line,arc, or polyline segment to bisect - select a line, an arc, a polyline segment, or a polyarc. • Line or polyline segment - draws xline perpendicular to the segment's midpoint: 

	• Arc or polyarc - draws xline perpendicular to the arc's center point and through its midpoint:  This option works with splined polylines, but not with spline entities.
Vertex point	Draws xlines that start at the vertex point and bisect two pick points; prompts you: Bisection angle start point - specify one part of the bisection angle. Bisection angle start point - specify the other part of the bisection angle. 
Horizontal	Draws xlines horizontal to the current UCS; prompts you: Location - specify the horizontal location of the xline; pick a point in the drawing, or enter x,y coordinates. The 'Location' prompt repeats for drawing multiple horizontal xlines at different starting points. Press Enter to exit the command.
Vertical	Draws xlines vertical to the current UCS; prompts you: Location - specify the vertical location of the xline; pick a point in the drawing, or enter x,y coordinates. The 'Location' prompt repeats for drawing multiple vertical xlines at different starting points. Press Enter to exit the command.
Angle	Draws xlines at angles; prompts you: Reference/<Enter angle (0)> - enter an option.
Reference	Draws xlines at an angle relative to another entity; prompts you: Select reference entity - choose a line. The line provides the reference (base) angle. Enter angle - specify the angle, measured relative to the entity. Location - specify the angle's start point; need not coincide with the entity's end points.

	 The 'Location' prompt repeats for drawing multiple infinite lines at the same angle.
Enter angle	Specifies the angle; enter an angle or pick two points to indicate the angle. Prompts you: Location - specify the starting point of the xline.  The 'Location' prompt repeats for drawing multiple infinite lines at the same angle.
Parallel	Draws xlines parallel to lines and polyline segments; prompts you: Through point/<Offset distance for parallel infinite line> (Through) - enter an option. This option works only with line segments.
Through point	Specifies the point through which to draw the xline; prompts you: Select entity for parallel infinite line - choose the entity to offset with a xline. Through the point - specify the point through which to draw the xline; this point also specifies the xline's start point. 
Offset distance for parallel	Specifies the offset distance. Enter a value or pick two points; prompts you: Select entity for parallel infinite line - choose the entity to offset with an xline. Side for parallel infinite line - specify the side on which to place the xline.

infinite line	 The xline's start point is parallel to the line's start point. The 'Select entity for parallel infinite line' prompt repeats for drawing multiple xlines at different offset distances.
Direction	Specifies the direction of the xline. Pick a point, or enter x,y coordinates.

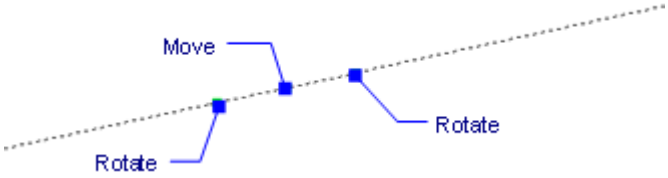
Procedures

Drawing infinite lines

Grips Editing

XLines can be edited directly through grips:

1. Select the xline. Notice that it has three grips.
2. Drag the grips to perform the following actions:
    - Drag the midpoint grip to move the xline.
    - Drag the other grips to rotate the xline.



Related Commands

Ray - draws semi-infinite lines.
Properties - modifies the properties of rays and xlines.

XOpen

Opens an externally-referenced drawing (Xref) for editing in a separate window.

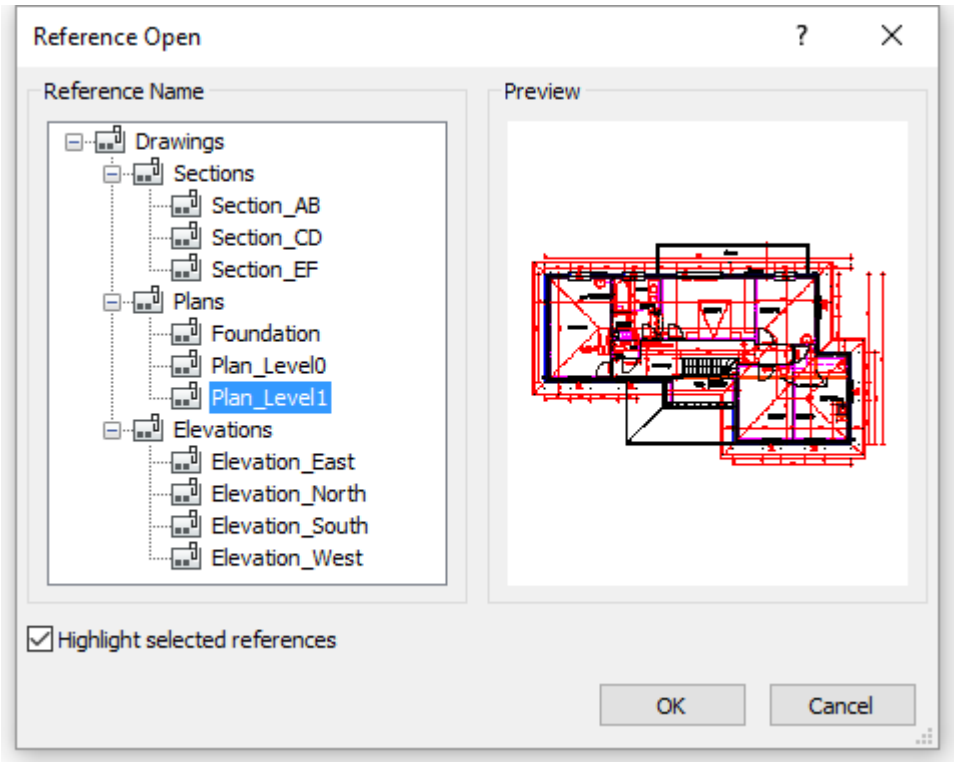
Accessing the Command

command bar: *xopen*
menu: *Xref context menu | Open Xref*
toolbar: *Drawing Explorer - External References |*

: ***xopen***
If launched from the command bar, prompts you:

Select reference: (Click an externally-referenced drawing in the current drawing)

If the selected externally-referenced drawing contains nested references a dialog box displays to let you choose the drawing to be opened:



Select the drawing you want to open and click the OK button.

Command Options

No Options

Related Commands

Refedit - edits block references and externally-referenced drawings through a dialog box.
BmOpen - opens the source drawing of an external mechanical component.

Xplode

Explodes entities, and provides control over the resulting entities (short for "explode").

Accessing the Command

command bar: *xplode*
transparent: *'xplode*
alias: *x*

: ***xplode***
Prompts you in the command bar:

Select entities to explode: (Choose one or more entities.)
Entities in set: 1

Select entities to explode: (Choose more entities, or press Enter to continue.)

Xplode entities [Separately/All] <All>: (Type S or A.)

[All listed/Multiple/LAyer/Color/LType/ltScale/Inherit all from parent] <Xplode>: (Enter an option.)

This command does not explode external references, attached images, or basic entities, such as lines, circles, and arcs.

Command Options

Option	Description
Select entities to explode	Selects the entities to explode. Choose one or more entities; use any selection method. Entities on frozen layers are not selected and not frozen.
Separately	Explodes one entity at a time; the prompts repeat themselves for each entity.
All	Explodes all selected entities together.
All listed	Allows to specify all properties for the selected entities.
Multiple	Allows to specify more than one property for the exploded entities; prompts you: **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - enter an option. Choose the properties one by one. E.g. if you want to specify the color and the linetype scale: **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - type C, then press Enter. Altering: Color **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - type LT, then press Enter. Altering: Color LTScale [LAyer/Color/LType/ltScale/Exit] <Xplode> - Press Enter to proceed; prompts you: Color for drawing new entities <BYLAYER> - specify a color name or number. Linetype scale for new entities <1> - specify the linetype scale factor.

Color	Specifies the color for the exploded entities; prompts you: Color for new entities - specify a color name or number.
LAyer	Specifies the layer for the exploded entities; prompts you: Layer for new entities - specify the name. The layer name must exist in the drawing.
LType	Specifies the linetype for the exploded entities; prompts you: Linetype for new entities - specify the name. The linetype name must exist in the drawing.
ltScale	Specifies the linetype scale for the exploded entities; prompts you: Linetype scale for new entities - specify the scale factor.
Inherit all from parent	Inherits the color, layer, and linetype scale from the source entities.
Xplode	Explodes the selected entities.

Related Commands

Explode - explodes complex entities into simpler ones.
TxtExp - explodes text into polylines.
U - undoes the effect of the explosion.

XRef

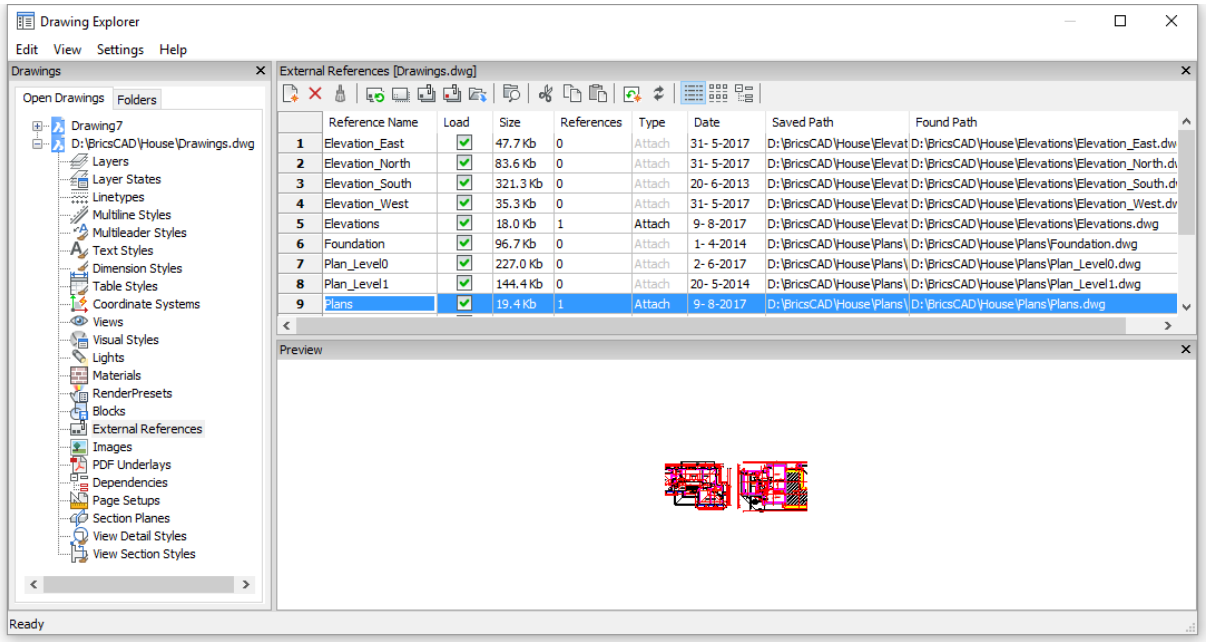
Attaches DWG files to the current drawing through the Drawing Explorer (short for "external reference").

Accessing the Command

command bar: *xref*
alias: *xr, xa, xattach, expxrefs*
menu bar: *Settings | XRefs*

: ***xref***

Displays Drawing Explorer



Attach a DWG file, or change the properties of existing xrefs, and then click ***X***.

Press F1 to access help.

The XDWGFADECTL system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Command Options

Option	Description
Reference Name	Specifies the name of the xref. This name is assigned by BricsCAD, but can be changed by the user. Click to rename.
Loaded	Toggles the loading of the xref: • On - loads the xref, and makes it visible. • Off - unloads the xref, making it invisible. Click to change the load status.

Size	Reports the size of the xref DWG files. Very large files can slow down the system.
References	Reports the number of times the xref is attached to the drawing.
Type	Reports whether the xref is attached or overlaid: • Attach - all xrefs are displayed. • Overlay - only the first xref of nested xrefs is displayed. To change type, you must unload the xref and then reload it as attached or overlaid.
Date	Reports the date of the file. This can be useful in determining if you ware working with the most recent revision.
Saved Path	Reports the original path to the xref's DWG file when first loaded. **Saved Path** | **Found Path** \\Ralph\d\CAD\dwg\LTSAM | \\Ralph\d\CAD\dwg\LTS xref4 | (! Not found) If the path cannot be found, click this field, and then click the button. BricsCAD displays the Choose a File dialog box" 1. Locate the missing DWG file. 2. Click Open. To convert the Xref path type, do the following: 1. Right click the Xref name. 2. Choose the path type in the context menu: Absolute path Filename as a path or Relative path Filename as a path
Found Path	Reports the current path to the xref's DWG file. This path should match the Saved Path in most cases.
Attach Xref	Attaches DWG files as xrefs; prompts you in the command bar: External References: ? to list/Bind/Insert/Detach/Path/Reload/Unload/Overlay/<Attach> - **enter an option.** See **-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 Loaded column.
Bind Xref	Binds the xref to the current drawing; Binds the xref; makes the xref part of the drawing. References to the xref disappear from the Drawing Explorer. This option is not available when xrefs are unloaded. This option sets the BINDTYPE settings 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. This option is not available when xrefs are unloaded.
Open Xref	Opens the Xref for editing.
Detail View	Displays the details of xrefs: [illegible]
Icon View	Displays icons of the xrefs: [illegible]
Tree View	Displays the relationship between xrefs in a tree-like view: [illegible] Current - current drawing. Column and Floor - xrefs attached to the current drawing.

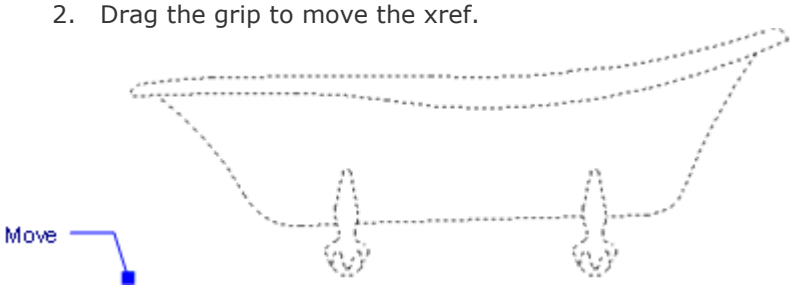
Overlay and Attach operate the same way. The difference is in how they treat xrefs that have xrefs themselves"

- Overlay attaches only the first level of xref.
- Attach attaches all xrefs.

Grips Editing

External references can be edited directly through grips:

1. Select the xref. Notice that it has one grip at its insertion point.
2. Drag the grip to move the xref.



Procedures

Managing Xrefs
To detach external references
To unload external references
To reload external references

Related Commands

Insert - inserts drawings as blocks.
Eplorer - opens the Drawing Explorer dialog box
PdfAttach - inserts a PDF file as an underlay into the current drawing.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
Pdfclip - allows clipping of PDF underlays.
Pdflayers - allows to control the display of layers in a PDF underlay.
Pdf - attaches PDF files to the current drawing through the Drawing Explorer.
PlaceView - places a model view selected from a source drawing into a paper space layout of the current drawing.
-XRe f- attaches and controls externally referenced drawings through the command bar.
XAttach - attaches xrefs through dialog boxes.
XClip - clips externally-referenced drawings.
VpClip - clips viewports.

-XRef

Attaches DWG files to the current drawing through the command bar (short for "external reference").

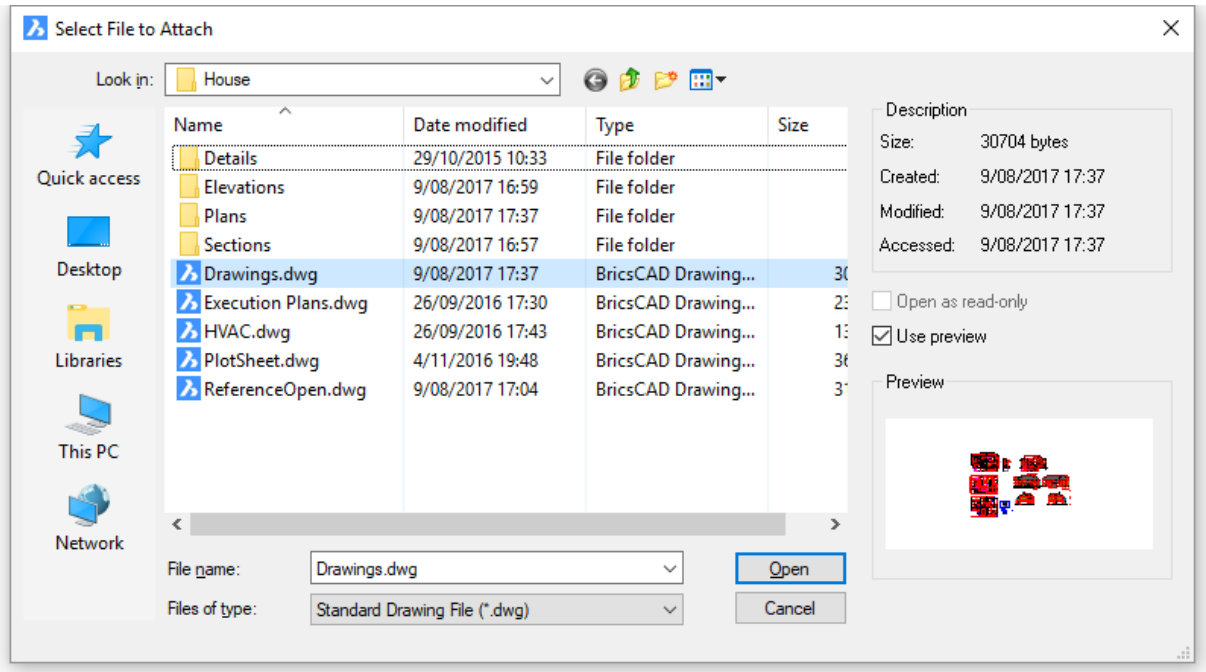
Accessing the Command

command bar: *-xref*
alias: *-xr*

: ***-xref***
Prompts you in the command bar:

External References [? to list external references/Attach/Overlay/Bind/Detach/Path/Reload/Unload] <Attach>: (Enter an option, or press Enter to attach drawing.)

Displays a dialog box:



Choose a DWG file, and then click Open.

BricsCAD prompts you in the command bar (see -Insert command):

Insertion point for block: (Pick a point, or enter x,y coordinates.)

Corner/XYZ/X scale factor <1.000000>: (Enter an option, a number, or press Enter for the default value.)

Y scale factor: < Equal to X scale (1.000000)>: (Enter an option, a number, or press Enter for the default value.)

Rotation angle for block <0>: (Enter a number, or press Enter for the default value.)

The XDWGFADECTL system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Command Options

Option	Description
? to list	Lists the names of all blocks in the current drawing. Prompts you: Xref(s) to list <* for all> - press Enter to list xrefs already attached, or enter part of a name to view some xrefs. Sample report: **Xref name Xref Type Path** **--- --- ---** **"XREF1" Attach d:\CAD\dwg\XREF1\.DWG** **Total Xref(s): 1** If necessary, press F2 to view the full list. See **TextScr** command.
Bind	Depending on the BINDTYPE settings variable: If BINDTYPE = ON (Insert-like behavior): Converts xrefs into blocks; prompts you: Externally referenced drawing(s) to insert - enter the names of one or more xrefs. Separate multiple names with commas. If BINDTYPE = OFF (Traditional binding behavior): Binds the xref; makes the xref part of the drawing; prompts you: Externally referenced drawing(s) to bind - enter the names of one or more xrefs. Separate multiple names with commas.
Insert	This option is hidden, but you can type *i* or *insert* at the command prompt. Converts xrefs into blocks; prompts you: Externally referenced drawing(s) to insert - enter the names of one or more xrefs. Separate multiple names with commas.
Detach	Detaches xrefs from the drawing; prompts you: Externally referenced drawing(s) to detach - enter the names of one or more xrefs. Separate multiple names with commas. Detached xrefs are removed from the drawing, and are no longer displayed. Use the Attach option to reattach them.
Path	Changes the path to the xref's source DWG file; prompts you: Externally referenced drawing(s) to edit path - enter the names of one or more xrefs. Displays the New Location for XREF dialog box. Choose a new path, and then click Open. Separate multiple names with commas; the New Location for XREF dialog box repeats for each xref named. When the path is incorrect, BricsCAD cannot display the xref. Use this option when the source DWG file is located in a different folder or on another computer.

Reload	Reloads the xrefs from their source DWG files; prompts you: Externally referenced drawing(s) to reload - enter the names of one or more xrefs. BricsCAD reports whether the DWG file has changed since the xref was last loaded: -- Reload Xref XREF1: d:\CAD\XREF1.DWG -- XREF1 has not changed 1 reference updated Separate multiple names with commas. Use this option when the source DWG file has changed, and you want to see the updated drawing.
Unload	Unloads the xrefs from the drawing; prompts you: Externally referenced drawing(s) to unload - enter the names of one or more xrefs. BricsCAD reports the xrefs that were unloaded: -- Xref XREF2: d:\CAD\XREF2.DWG -- XREF2 is unloaded Separate multiple names with commas. This option keeps the links to the source DWG files, and hides the display of xrefs in the drawing.
Overlay	Overlays xrefs in the current drawing; displays the Select File to Overlay dialog box. Choose a DWG file, and then click Open. BricsCAD prompts you in the command bar: Insertion point for block - pick a point, or enter x,y coordinates. Corner/XYZ/X scale factor - enter an option, a number, or press Enter for the default value. Y scale factor: < Equal to X scale> - enter an option, a number, or press Enter for the default value. Rotation angle for block - enter a number, or press Enter for the default value. See Insert command.
Attach	Attaches xrefs to the current drawing; displays the Select File to Attach dialog box. Choose a DWG file, and then click Open. BricsCAD prompts you in the command bar: Insertion point for block - pick a point, or enter x,y coordinates. Corner/XYZ/X scale factor - enter an option, a number, or press Enter for the default value. Y scale factor: < Equal to X scale> - enter an option, a number, or press Enter for the default value. Rotation angle for block - enter a number, or press Enter for the default value. See Insert command.

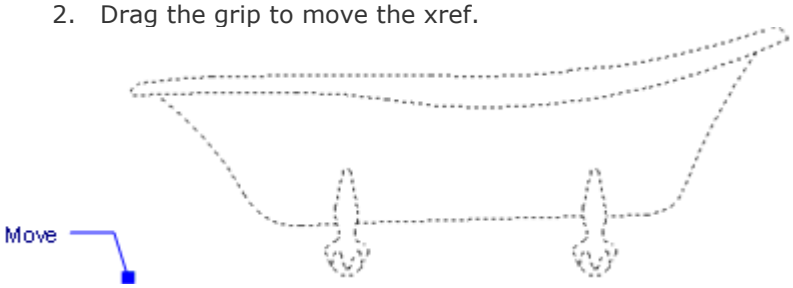
Overlay and Attach operate the same way. The difference is in how they treat xrefs that have xrefs themselves (nested xrefs).

- Overlay attaches only the first level of xref, nested xrefs are not attached.
- Attach attaches all xrefs, including nested xrefs.

Grips Editing

External references can be edited directly through grips:

1. Select the xref. Notice that it has one grip at its insertion point.
2. Drag the grip to move the xref.



Related Commands

pdfattach - inserts a PDF file as an underlay into the current drawing.
-PdfAttach - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.
XRef - attaches and controls externally referenced drawings through the Drawing Explorer.
XAttach - attaches xrefs through a dialog box.
XClip - clips externally-referenced drawings.
Insert - inserts drawings as blocks.

Z

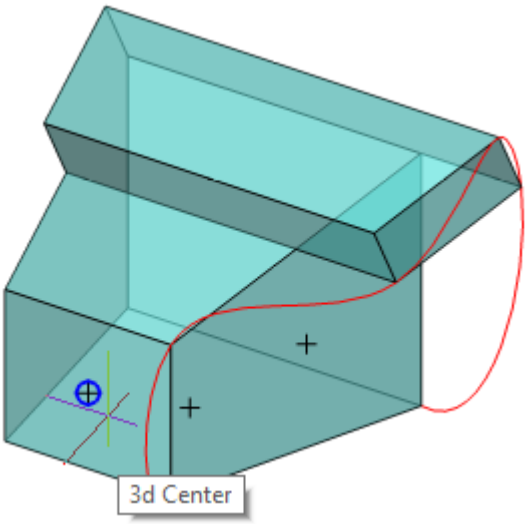
Zcenter

Toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.

Accessing the Command

command bar: *zcenter*
transparent: *zcenter*
toolbar: *3D Entity Snaps |*

: ***zcenter***
Displays no prompts in the command bar.



Command Options

Option	Description
zcenter	This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('). This can be expensive on complex curved faces. Note that activating the 2D Geometric Center snap is sufficient to snap to the center of planar faces.

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
GCenter - toggles Geometric Center (GCenter) entity snap; snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.

Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

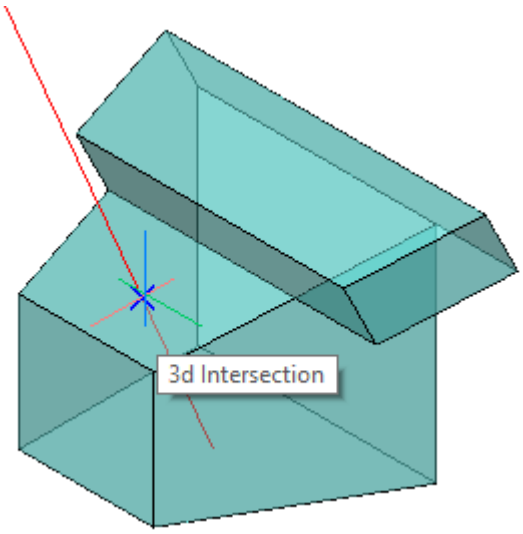
Zintersection

Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.

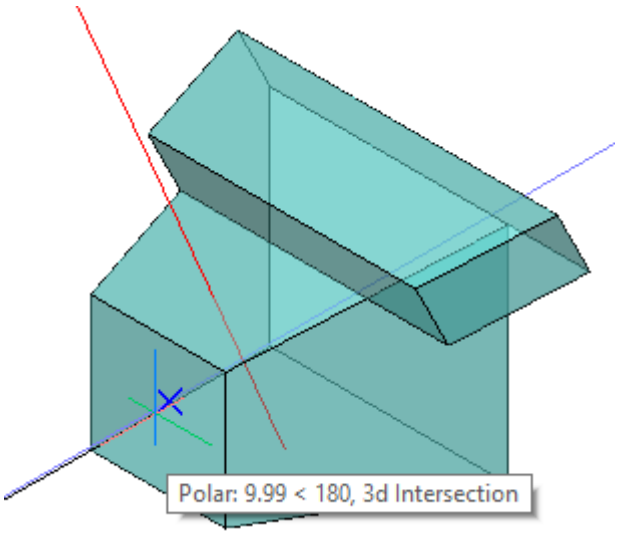
Accessing the Command

command bar: *zintersection*
transparent: *zint*
toolbar: *3D Entity Snaps |*

: ***zintersection***
Displays no prompts in the command bar.



Snap to the intersection point of a line and the face of a 3d solid.



Snap to the intersection point of a Polar tracking line and the face of a 3d solid.

Command Options

Option	Description
zint	This keyword can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
Midpoint - toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and edges of 3D solids.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

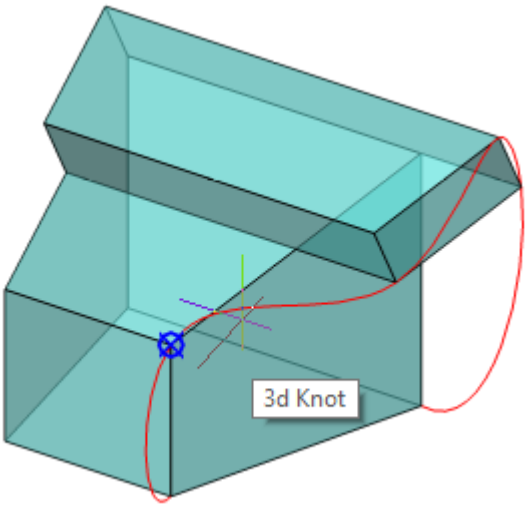
Zknot

Toggles the 3D knot entity snap. Snaps to a knot on a spline.

Accessing the Command

command bar: *zknot*
transparent: *zknot*
toolbar: *3D Entity Snaps |*

: ***zknot***
Displays no prompts in the command bar.



Command Options

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

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

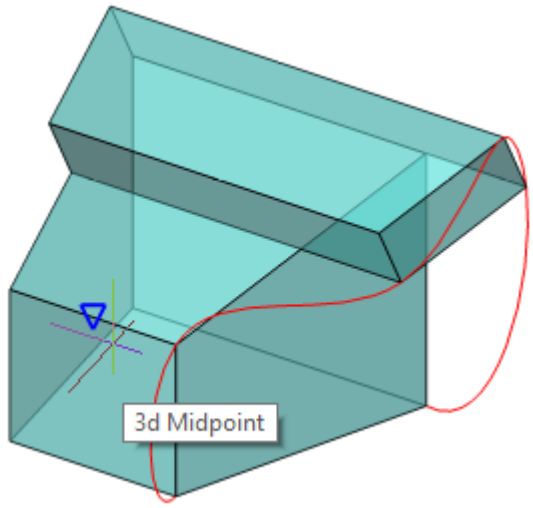
Zmidpoint

Toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.

Accessing the Command

command bar: *zmidpoint*
transparent: *zmidpoint*
toolbar: *3D Entity Snaps |*

: ***zmidpoint***
Displays no prompts in the command bar.



Command Options

Option	Description
zmidpoint	This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('). If 3D snapping (Zmidpoint) is off the 2D snap (Midpoint) will still work on 3D objects. If 2D snapping is off then 3D snap (for midpoints)will not work on 2D objects, so the 3D snap is a '3D only' snap.

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
Midpoint - toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and edges of 3D solids.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

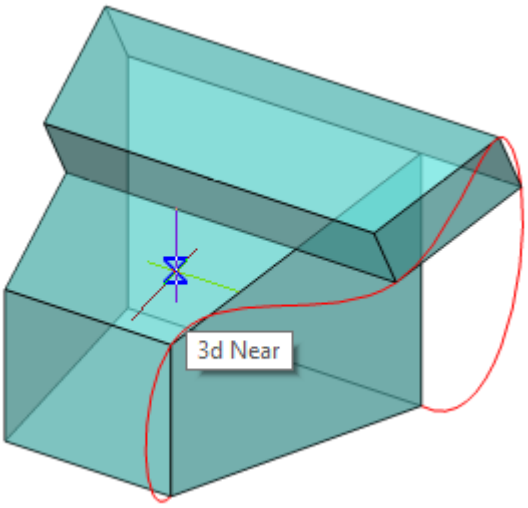
Znearest

Toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.

Accessing the Command

command bar: *znearest*
transparent: *znearest*
toolbar: *3D Entity Snaps |*

: ***znearest***
Displays no prompts in the command bar.



Command Options

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

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
Nearest - toggles Nearest entity snap mode; snaps to the nearest geometry on entities.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Znone

Disables all 3d snap modes.

Accessing the Command

command bar: *znone*
transparent: *znone*
toolbar: *3D Entity Snaps |*

: ***znone***
Displays no prompts in the command bar.

Command Options

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

Related Commands

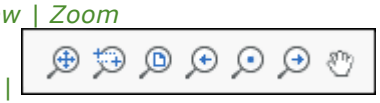
3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Zoom

Visually changes the size of the drawing within the current viewport.

Accessing the Command

command bar: *zoom*
transparent: *'zoom*
alias: *z*
menu bar: *View | Zoom*
toolbar: *Zoom |* 

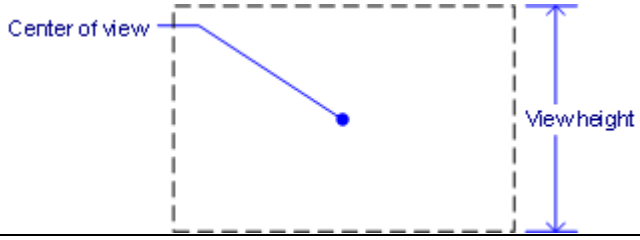
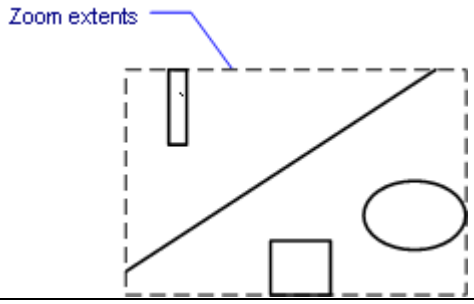
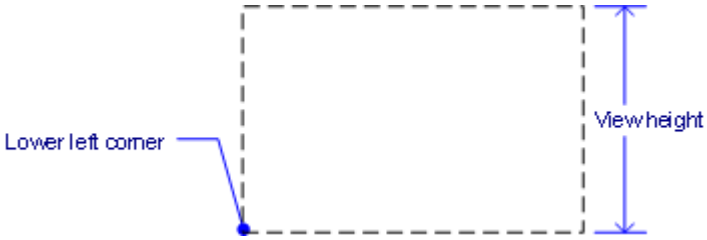
: ***zoom***
Prompts you in the command bar:

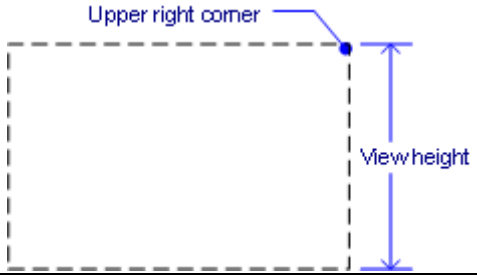
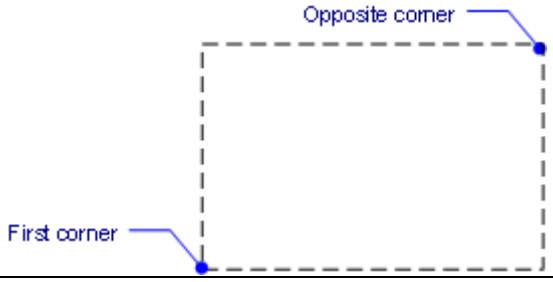
Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/Window/OBject] <Scale (nX/nXP)>: (Enter an option, or pick two points for a windowed zoom, or enter a zoom factor.)

To zoom in and out at any time, roll the mouse's wheel back and forth.

Command Options

Option	Description
Scale	Specifies the absolute zoom scale factor: • Less than 1 - makes the drawing appear smaller. • 1 - keeps the drawing the same size. • More than 1 - makes the drawing appear larger.
nX	Specifies the zoom factor relative to the current zoom. For example, 2x makes the current view twice as large.
nXP	Specifies the zoom factor in paper space relative to model space. For example, 2xp makes the model space view twice as large as the surrounding paper space.
In	Zooms in by 50% (0.5)
Out	Zooms out by 50% (2.0).
All	Displays the entire drawing; zooms to the larger of: • Drawing limits; see Limits command. • Extents of drawing, excluding entities on frozen layers.

Center	Zooms in or out about a specified center point; prompts you: Center of view - pick a point or enter x,y coordinates. Enter zoom factor (nX/nXP), or view height - enter a zoom factor or view height. The view height defines the zoom; it is the height of the current viewport. 
Dynamic	Pans and zooms using a rectangular box which represents your view. When you choose the *Dynamic* option, a view extents or view limits is executed first. The original size of the view box matches the size of the previous view. A dashed rectangle indicates the area of the previous view. You can adjust the size of the view box dynamically, then move it to the part of the drawing you want to view. A x in the center of the view box indicates the positioning mode. A right pointing arrow indicates the resizing mode. Click to switch between the two modes. • To adjust the size of the view box: Click to start resizing. Move the mouse to the right to enlarge, to the left to shrink. Click again to stop resizing the view box. • To zoom in: Move the view box to the part of the drawing you want to enlarge, then press Enter.
Extents	Displays the drawing to the extents of entities; limits and frozen entities are ignored. 
Left	Zooms in or out with the lower left corner specified by a point; prompts you: Lower left corner - pick a point or enter x,y coordinates. Enter zoom factor (nX/nXP), or view height - enter a zoom factor or view height. The view height defines the zoom; it is the height of the current viewport. 
Previous	Displays the previous view, whether set by the Zoom, Pan, or View commands.

Right	Zooms in or out with the upper right corner specified by a point; prompts you: Upper right corner - pick a point or enter x,y coordinates. Enter zoom factor (nX/nXP), or view height - enter a zoom factor or view height. The view height defines the zoom; it is the height of the current viewport. 
Window	Zooms in to a rectangular area; prompts you: First corner - pick a point or enter x,y coordinates. Opposite corner - pick another point or enter x,y coordinates. 
Object	Allows to zoom in on a single entity or a selection set; prompts you: Select an entity - select one or more entities. Entities in set: 1 Select an entity - select more entities or press Enter to zoom in on the selection set.

Related Commands

RtZoom - performs real-time zooming of the drawing.
Pan and -Pan - move the drawing within the current viewport.
RtPan - performs real-time panning of the drawing.

View and -View - set up named views of drawings.
DdVPoint - changes the 3D viewpoint of the drawing through a dialog box.
VPoint - changes the 3D viewpoint of the drawing through the command bar.
RtRot - rotates the 3D viewpoint in realtime.
RtRotX - rotates the 3D viewpoint about the x axis in realtime.
RtRotY - rotates the 3D viewpoint about the y axis in realtime.
RtRotZ - rotates the 3D viewpoint about the z axis in realtime.

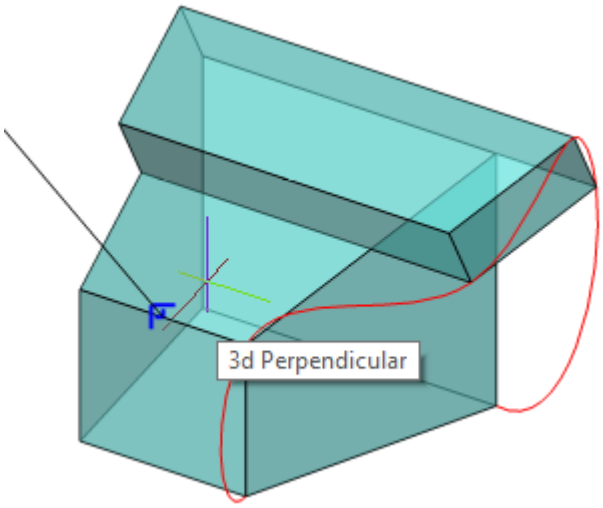
Zperpendicular

Toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.

Accessing the Command

command bar: *zperpendicular*
transparent: *zperpendicular*
toolbar: *3D Entity Snaps |*

: ***zperpendicular***
Displays no prompts in the command bar.



Command Options

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

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
Perpendicular - toggles perpendicular entity snap mode.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zvertex - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

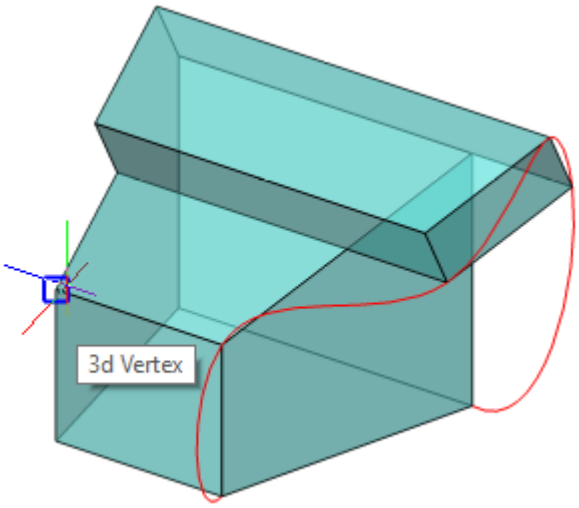
Zvertex

Toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Accessing the Command

command bar: *zvertex*
transparent: *zvertex*
toolbar: *3D Entity Snaps |*

: ***zvertex***
Displays no prompts in the command bar.



Command Options

Option	Description
zvertex	This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('). If 3D snapping (Zvertex) is off the 2D snap (Endpoint) will still work on 3D objects. If 2D snapping is off then 3D snap (for endpoints)will not work on 2D objects, so the 3D snap is a '3D only' snap.

Related Commands

3dOsnap - sets the entity snap modes for 3D entities through the Settings dialog box.
-3dOsnap - toggles the entity snap modes for 3D entities at the command line.
OSnap - sets entity snaps through the Settings dialog box.
-OSnap - toggles entity snap modes at the command line .
Aperture - sizes the aperture cursor for entity snaps.
Endpoint - toggles endpoint entity snap; snaps to the ends of open entities, such as line, arcs, and open polylines and vertices of 3D solids.

Zcenter - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces.
Zintersection - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces.
Zknot - toggles the 3D knot entity snap. Snaps to a knot on a spline.
Zmidpoint - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.
Znearest - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.
Znone - disables all 3d snap modes.
Zperpendicular - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.