Bricscad V8 – User Guide

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Bricscad

Welcome to Bricscad

Congratulations on your purchase of Bricscad. Whether you're a novice user or an experienced CAD professional, you'll soon be creating drawings using the numerous productivity-enhancing features in the software. These features include:

- A familiar Windows environment.
- The ability to work with multiple open documents.
- Unparalleled DWG compatibility.

These are just a few of the many features that we have included in the program to make the transition into CAD-based drawing a smooth one for new users. We have also incorporated many features that experienced CAD users have been requesting for years in a drawing package. You have chosen an affordable, high-quality software program to produce your drawings. We are conviced you will be pleased with the results!

We encourage you to take a few moments to familiarize yourself with the information in this Help system. We have organized the following quick-reference topics to give you an overview of some of the program's features and to assist you in using the Bricscad Help system.

Because Bricscad can read, write, and display DWG files without conversion, it is an obvious choice for Autodesk® AutoCAD users. But the program provides other compatibility and productivity features that you will find indispensable.

Bricscad is a powerful drawing program that gives you the ability to create professional two-dimensional drawings and three-dimensional designs. But don't take our word for it, see a few samples for yourself!

Bricscad is designed for anyone who wants a fast and efficient CAD program with the power and versatility of standard programs such as Autodesk® AutoCAD® or MicroStation® by Bentley Systems Inc., at an affordable price. Using today's advanced technology, Bricscad integrates the Microsoft® Windows® interface with a powerful CAD engine.

Understanding Bricscad

Bricscad is designed for anyone who wants a fast and efficient CAD program with all the power and versatility of standard programs such as Autodesk® AutoCAD® , or MicroStation® by Bentley Systems®, at an affordable price. Using today's advanced technology, Bricscad integrates the Microsoft® Windows interface with a powerful CAD engine.

Bricscad provides unparalleled compatibility with AutoCAD®, using most of the same file formats including those for drawings (.dwg files), linetypes, hatch patterns, and text styles. You can also use AutoCAD® menu files and run Autodesk® AutoLISP® programs. If you have written your own ADS (AutoCAD Development System® by Autodesk®) programs, simply recompile them to link with the Bricscad libraries. Many third-party ADS programs already support Bricscad. If you have a program that is not already supported, ask your software vendor to provide a Bricscad-compatible version of their program.

Bricscad is more compatible with the AutoCAD® program than any other CAD product, delivers additional tools with advanced CAD features, and has a seamless Microsoft® Windows® integration. This powerful program provides a superb combination of features for CAD users like architects, engineers, and designers.

Bricscad incorporates all the standard features found in other CAD programs, along with features and capabilities you won't find anywhere else. Its multiple-document interface (MDI) lets you open and work with several drawings at the same time. You can easily copy drawing entities between drawings. In addition, the powerful Bricscad Explorer lets you manage information and settings and quickly copy layers, linetypes, and other information between drawings. The brand new Settings Dialog lets you manage all settings variables in a single window. With the powerful search tool you can find any setting within seconds.

Installation platforms and levels

Depending on the license you have, Bricscad can be installed on 2 different platforms : MS-Windows or Linux.

For the Windows version you have the choice between 2 different levels : Classic or Pro.

Differences between Bricscad for Linux, Bricscad Classic and Pro:

Feature	Linux	Windows	
		Classic	Pro
Component Object Model (COM)	⭕	✔	✔
True Color	✘	✔	✔
Apparent & Extended Intersection Snap	✘	✔	✔
SnapTrack	✘	✔	✔
eDrop	✘	✔	✔
ActiveX, Including in-place-editing	✘	✔	✔
LISP encryption	✘	✔	✔
Quick Render (future release)	✘	✔	✔
Raster Image Support	✘	✔	✔
ACIS Viewing	✘	✔	✔
ACIS Editing	✘	✘	✔
Visual Basic for Applications (VBA)	✘	✘	✔

⭕ partly available

✘ not available

✔ available

This help file describes all the features. So some of the features do not apply to the Linux or Classic version.

NOTE In Evaluation mode (30 days trial period) Bricscad Pro is installed as a Classic version because the Pro features are not installed with a trial license key. After the trial period you need to run the installation again in order to install a Pro license key and the specific Pro features: VBA and ACIS Editing.

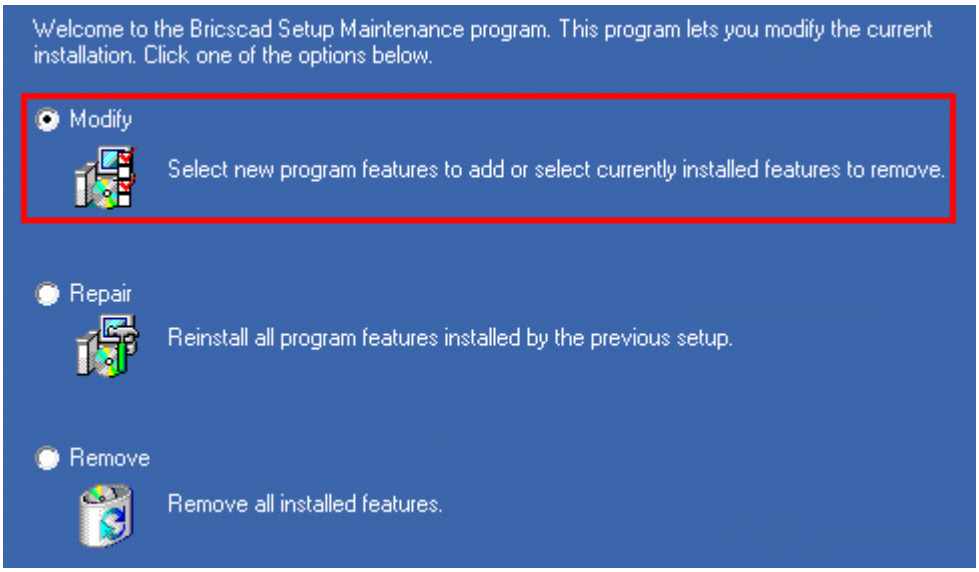
Upgrading a Classic version to a Pro version

It is possible to install a *Bricscad Pro* version without uninstalling an existing *Classic* version. You only need a *Bricscad Pro* installer (or the installer from a Bricscad CD) and a valid *Pro license key*.

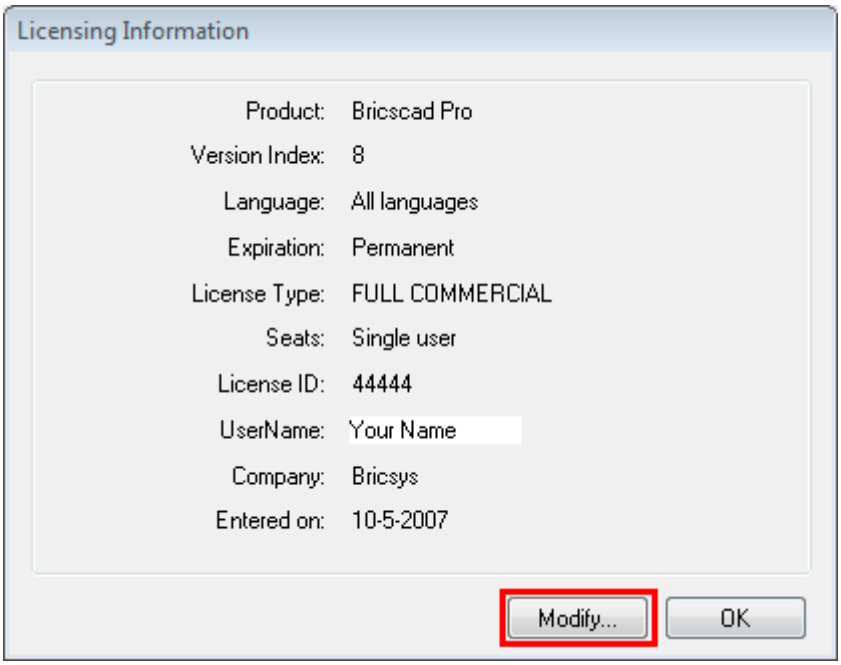
NOTE A valid Pro License key must be entered before or during the installation (and not after). This because the licensed features are installed only if they are authorized by the active licence key !

To upgrade an installed Classic version to the Pro version

1. Run the *Bricscad Pro* installer (or the installer from a Bricscad CD).
2. Select the *Modify* option.

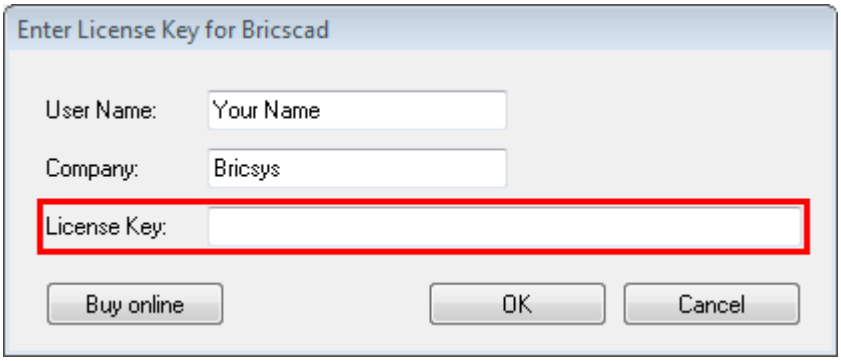


3. Click the *Next* button.
4. Accept the license agreement, then press the *Next* button.
5. In the *Licensing Information* dialog box, click the *Modify...* button.



6. Enter the new Pro licence key in the *License Key* field to replace the already installed Classic license key.

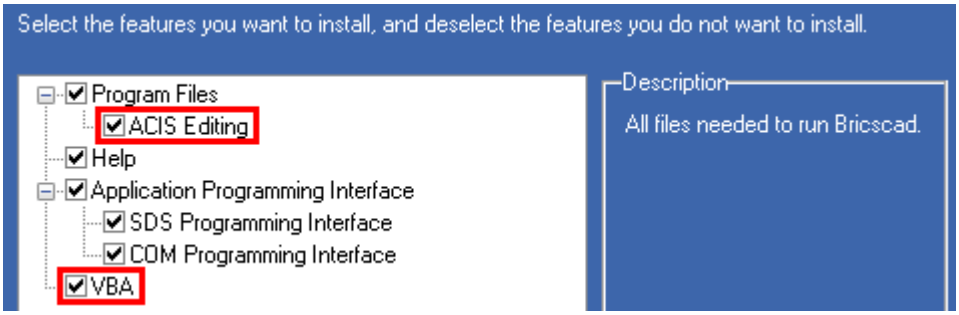
If the licence key was sent to you by mail, it is recommended to copy/paste the key from the email message into the *License Key* field.



7. Press the *OK* button to confirm the license key.

8. Press the *OK* button on the *License Information* dialog box.

9. Select the Pro features: *ACIS Editing* and *VBA* .



These features are only selected if a Pro licence key was previously installed.

Do not unselect other (installed) features : unselect a feature means uninstall it !

7. Click the *Next* button to finalize the installation.

The Pro features VBA (menu Tools > Visual Basic... > Visual Basic Editor.) and ACIS Editing (menu View > Toolbars > Solids Editing) are available now .

NOTE In Evaluation mode (30 days trial period) Bricscad Pro is installed as a Classic version because the Pro features are not installed with a trial license key. After the trial period you need to run the installation again in order to install a Pro license key and the specific Pro features: VBA and ACIS Editing.

Silent Installation (MS-Windows only)

Silent installations are installations that run without a user interface. A normal (non-silent) installation receives the necessary input from the user in the form of responses to dialog boxes. However, a silent installation does not prompt the end user for input. Instead, a silent installation must get its user input from a response file (.iss file).

A response file contains information that an end user would ordinarily enter as responses to dialog boxes when running a normal installation. During a silent installation (when the user runs Setup.exe with the /s switch), Setup.exe reads the necessary input from the response file at run time.

The format of response files resembles that of an .ini file, but response files have the .iss extension. A response file is a plain text file consisting of sections containing data entries.

Creating a response file

To create a response file, run Setup.exe with the /r switch, which runs the installation normally, and additionally creates the response file used by a silent installation. By default, the response file is called Setup.iss, and is created in the Windows or WinNT folder. To specify a different name or location for the response file, use the /f1"ISS file path" switch to Setup.exe.

Running a silent installation

To run an installation silently based on the contents of a response file, the end user runs Setup.exe with the /s switch in combination with the /f1"ISS file path" switch to specify the location of the response file.

A silent installation program does not display a dialog if an error occurs. Instead, status information for the silent installation is recorded (by default) in a file called Setup.log, created in the same folder as the response file being used. The end user can specify a different name and location for the log file using the /f2"LOG file path" switch to Setup.exe.

What's New

What's New

Compared to Bricscad V7 the following is new in V8:

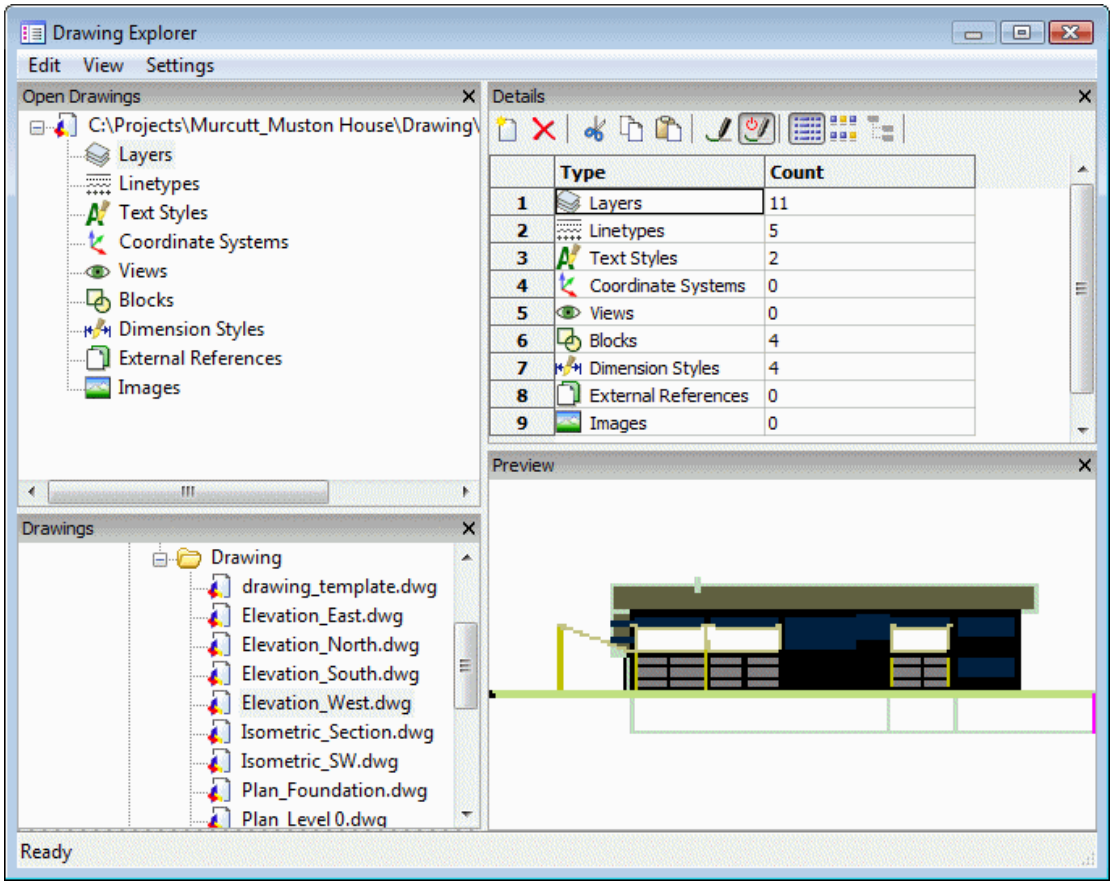
- Drawing Explorer
- Xref Explorer integrated in the Drawing Explorer
- Image Explorer integrated in the Drawing Explorer
- Properties bar
- Settings dialog

Drawing Explorer

Drawing Explorer

The new drawing explorer window consists of 4 sub-windows:

- *Open Drawings*: a list of all drawings that are currently open
- *Details*: the details of the selected drawing or the details of the selected category in a drawing, e.g. layers, blocks, images, ...
- *Drawings*: your favorite drawing folders
- *Preview*: a preview of the selected drawing or block



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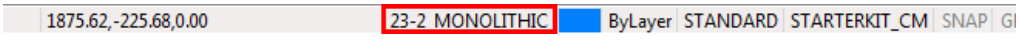
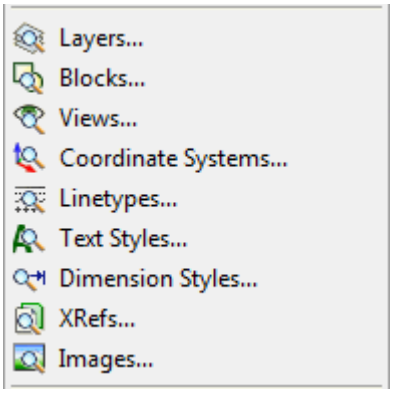
How to open the drawing explorer

To open the Drawing Explorer window to one of the following:

- Double click the *Layer field* in the *Status Bar*.
  The Drawing Explorer window opens showing the details of the *Layers* in the current drawing.



- Choose a *Settings Category* in the *Settings* menu.
  The Drawing Explorer window opens showing the details of the selected category.



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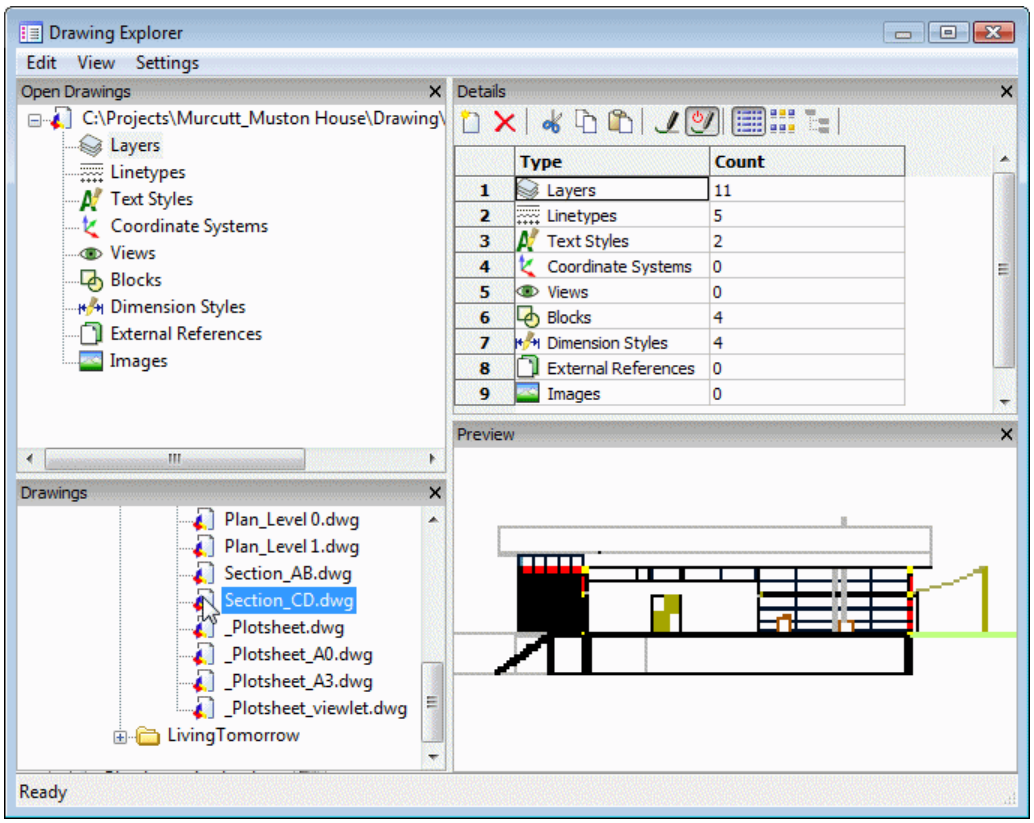
Opening a drawing

To open a drawing using the Drawing Explorer do the following

1. Launch the *Drawing Explorer*.
2. In the *Drawings* sub-window browse to the folder of the drawing.



3. Double click the drawing.

The drawing opens, while the *Drawing Explorer* window stays open.

4. (optional) Repeat steps 2 and 3 to open more drawings.
5. Close the *Drawing Explorer*.

NOTE When you select a drawing, a preview displays in the *Preview* sub-window.

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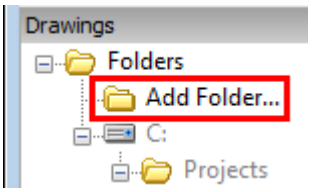
Adding a drawing folder

1. In the *Drawings* sub-window of the *Drawing Explorer* main window click *Add Folder...*



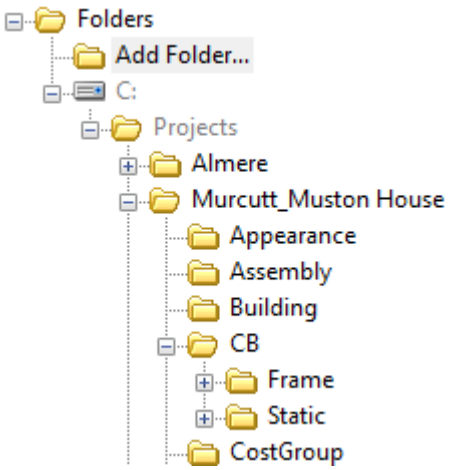
The *Browse For Folder* window opens.

2. In the *Browse For Folder* window do one of the following:

- To add an existing folder: select the folder you want to add.
- To add a new folder: click the Make New Folder button.

3. Click the *OK* button.

The folder is added.



NOTE The parent folder(s) of the selected folder is (are) greyed out in the drawing folder tree. Drawings in such folders cannot be accessed.

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Exploring Blocks

Exploring Blocks

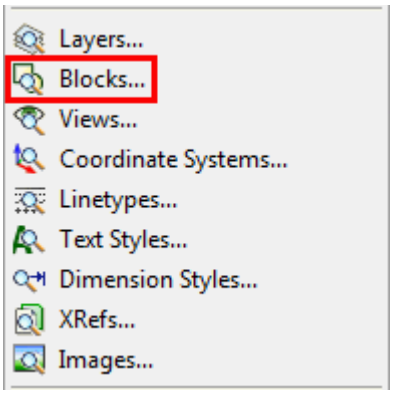
In the *Blocks Explorer* you can:

- Insert blocks ( ) in the current drawing
- Create new blocks ( )
- Delete blocks ( )
- Save a block as a (new) drawing ( )
- Insert Drawings as a block ( )
- Cut ( ) or Copy ( ) a block, then Paste ( ) the block in another drawing

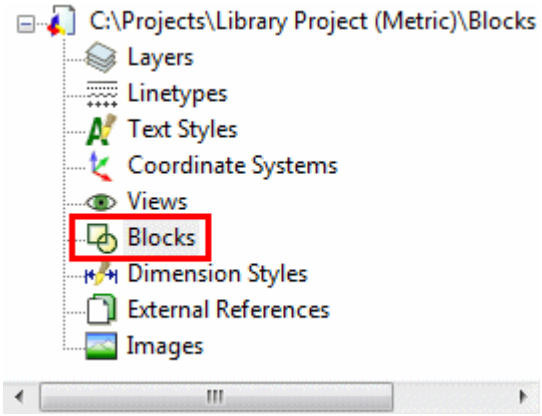
Open the Blocks Explorer

To open the *Blocks Explorer* do one of the following:

- Choose *Blocks* in the *Settings* menu.



- Select *Blocks* in the *Open Drawings* sub-window of the *Drawing Explorer*.



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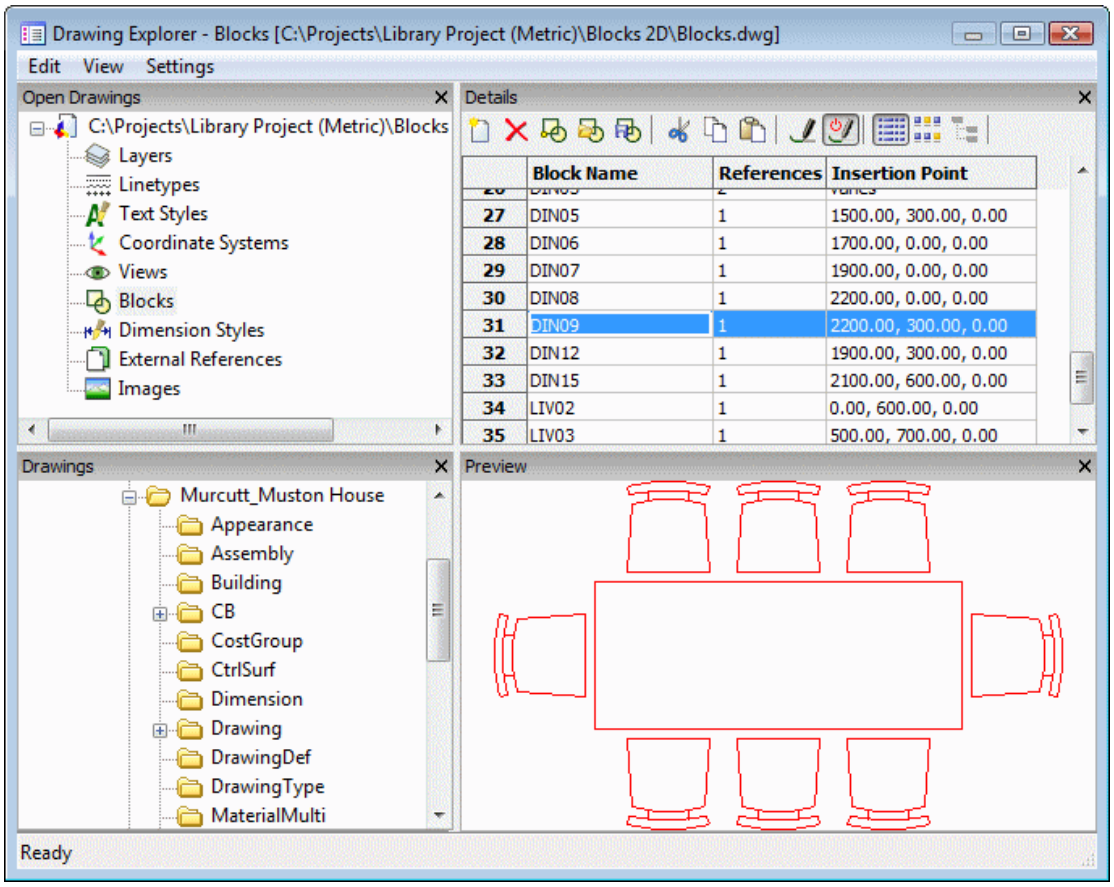
Blocks Explorer display options

You can choose between *Detail View* (list) and *Icon View* (thumbnails) to see the blocks in the current drawing.

Open the Detail View of the Blocks Explorer

Click the Detail View button () in the *Drawing Explorer* toolbar. The Detail View button is now pressed (), indicating the block details are displayed.
The selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



Detail View of the blocks in the current drawing

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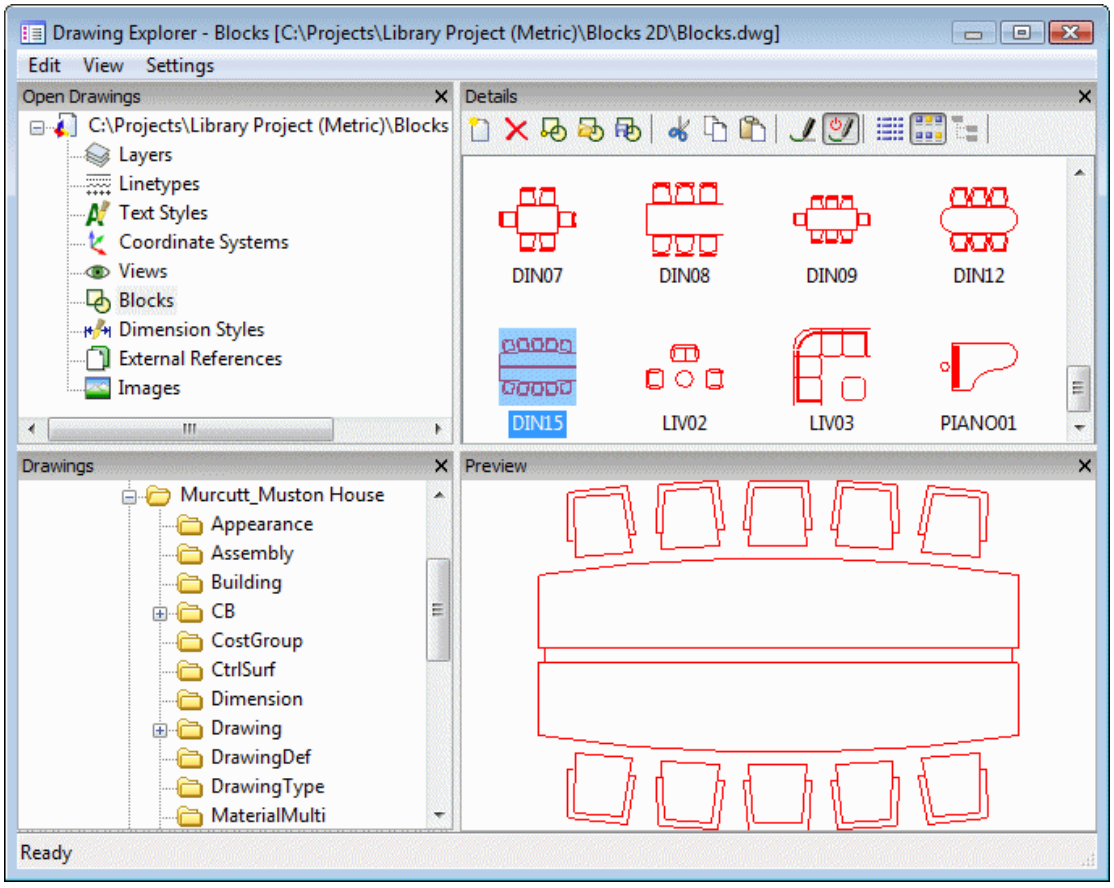
Open the Icon View of the Blocks Explorer

Click the Icon View button () in the *Drawing Explorer* toolbar. The Icon View button is now pressed (), indicating the block icons are displayed.
The selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



Icon View of the blocks in the current drawing

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Insert a block

1. Launch the *Block Explorer*.
2. (optional) Choose either *Detail View* ( ) or *Icon View* ( ).
3. Select the block.
4. Click the Insert Block button ( ) in the *Block Explorer toolbar*.
   The Block Explorer window closes.
5. Insert the block in the drawing.
   The Block Explorer windows opens.
6. (optional) Repeat steps 2 through 5 to insert more blocks.
7. Close the *Block Explorer* window.

NOTE Instead of clicking the Insert Block tool button in step 4 you can also:

- right click and select Insert from the context menu when in *Detail View*;

- double click the block when in *Icon View*.

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Exploring Xrefs

Exploring Xrefs

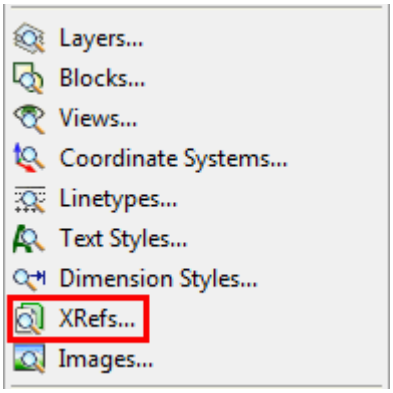
In the *Xrefs Explorer* you can:

- Attach Xrefs ()
- Detach Xrefs( )
- Reload Xrefs( )
- Unload Xrefs( )
- Bind Xrefs( )

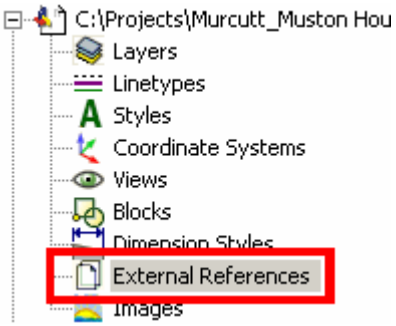
Open the Xrefs Explorer

To open the *Xrefs Explorer* do one of the following:

- Choose *Xrefs* in the *Settings* menu.



- .Select *External References* in the *Open Drawings* sub-window of the *Drawing Explorer*.



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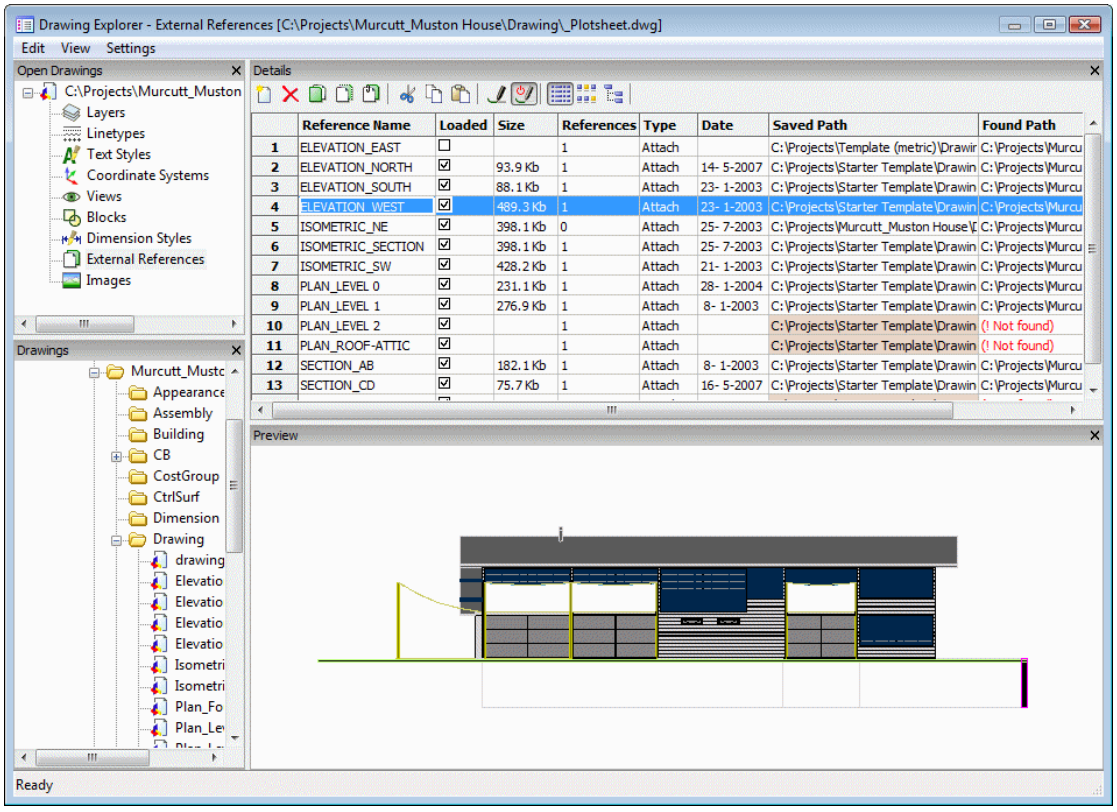
Xrefs Explorer display options

You can choose between *Detail View* (list), *Icon View* and *Tree View* to see the xrefs in the current drawing.

NOTE Bricscad searches for external references in the folder of the parent drawing first. If the drawing is not found there, Bricscad looks in the *Saved Path* folder. If the drawing is not found there either, (! Not Found) displays in the *Saved Path* field.

Open the Detail View of the Xrefs Explorer

Click the Detail View button ( ) in the *Drawing Explorer* toolbar. The Detail View button is now pressed ( ), indicating the Xref details are displayed.
The selected Xref displays in the *Preview* sub-window of the *Drawing Explorer*.



Detail view of the Xrefs in the current drawing

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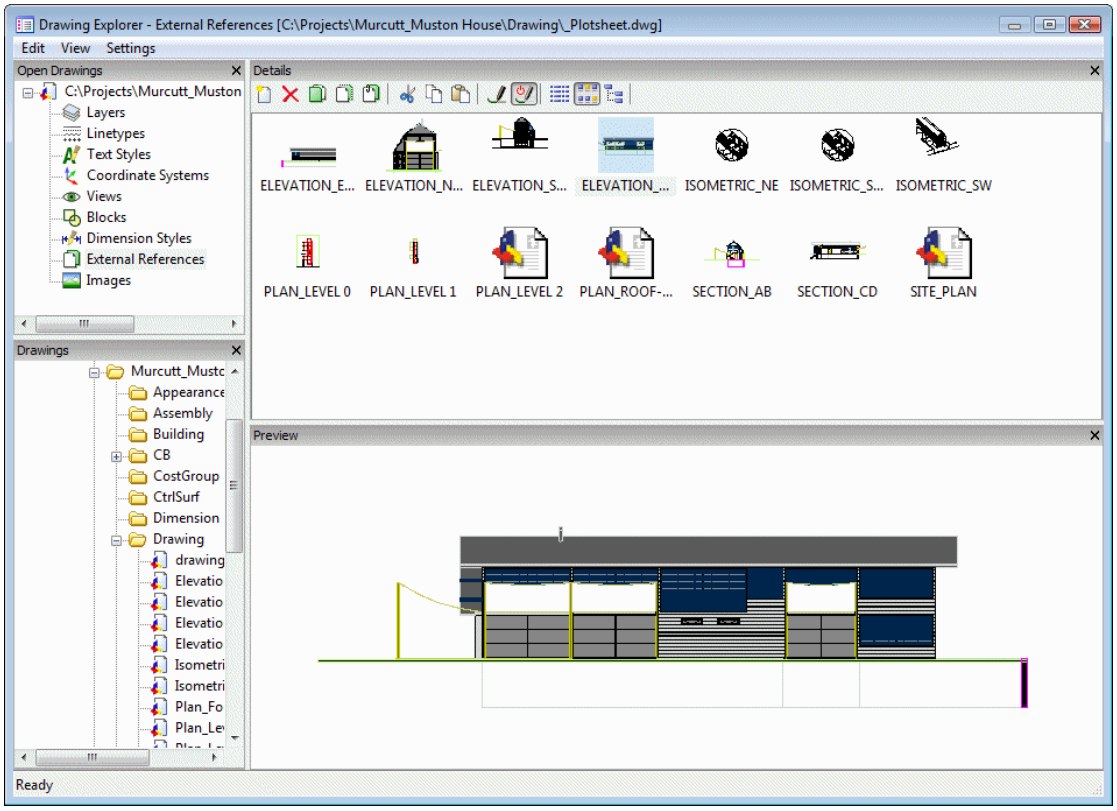
Open the Icon View of the Xrefs Explorer

Click the Icon View button ( ) in the *Drawing Explorer* toolbar. The Icon View button is now pressed ( ), indicating the Xref icons are displayed.
The selected Xref displays in the *Preview* sub-window of the *Drawing Explorer*.

External References that were not found display with the default icon ( ) in the *Details* sub-window of the *Drawing Explorer* window.



Icon view of the Xrefs in the current drawing

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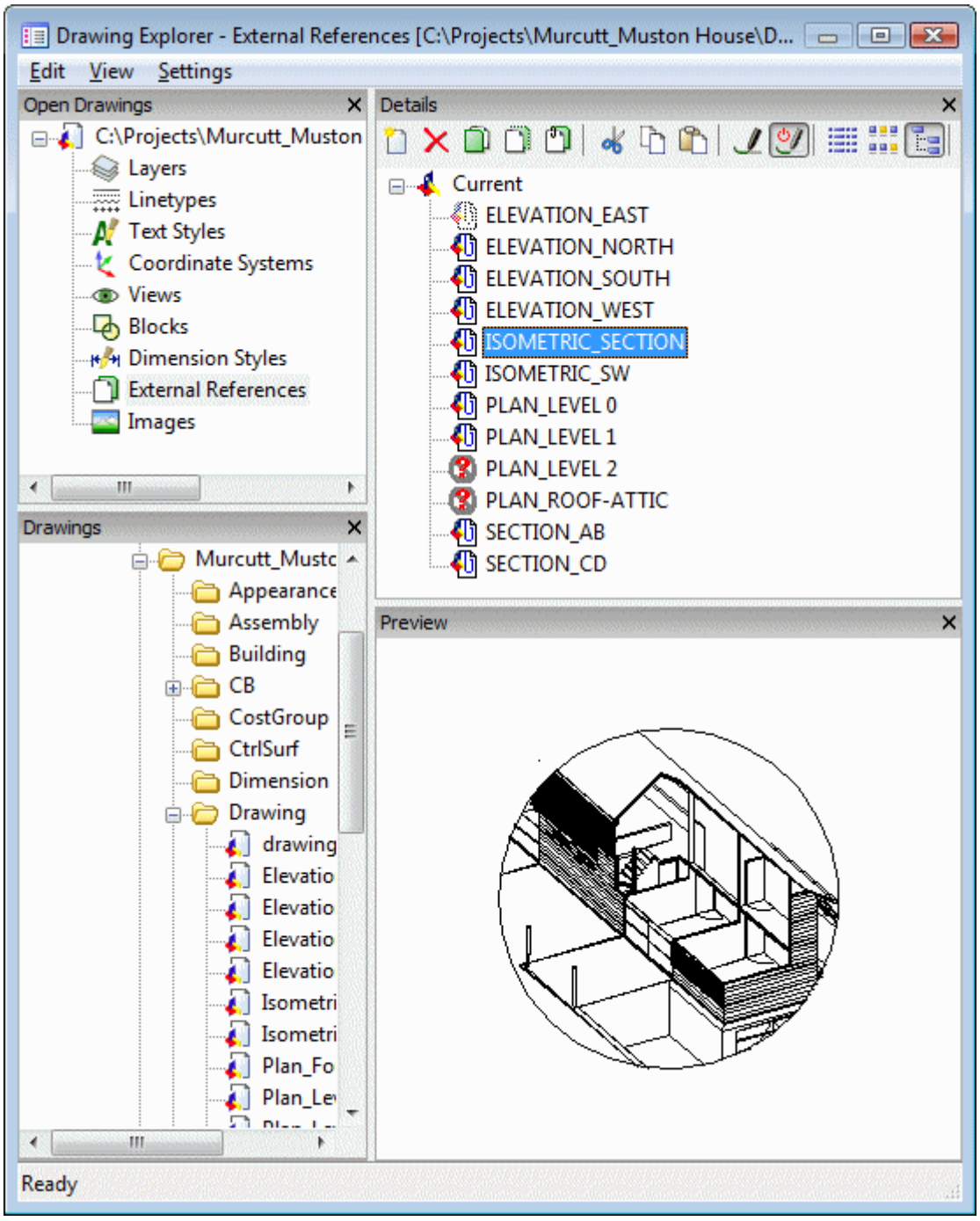
Open the Tree View of the Xrefs Explorer

Click the Tree View button () in the *Drawing Explorer* toolbar. The Tree View button is now pressed (), indicating the Xref tree is displayed.

The selected Xref displays in the *Preview* sub-window of the *Drawing Explorer*.

External References that were not found display with the error icon () in the *Details* sub-window of the *Drawing Explorer* window.



Tree view of the Xrefs in the current drawing

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Exploring Images

Exploring Images

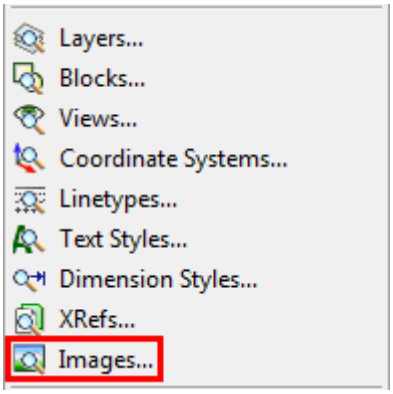
In the *Images Explorer* you can:

- Attach images ( )
- See a preview of the attached images
- Insert images ( )
- Load / Unload inserted images
- Detach images ( )

Open the Images Explorer

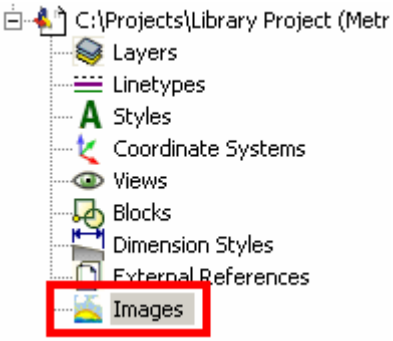
To open the Images Explorer do one of the following:

- Choose Images in the Settings menu.



- Click the Image Management tool button ( ) in the *Images* toolbar.

- Select *Images* in the *Open Drawings* sub-window of the *Drawing Explorer*.



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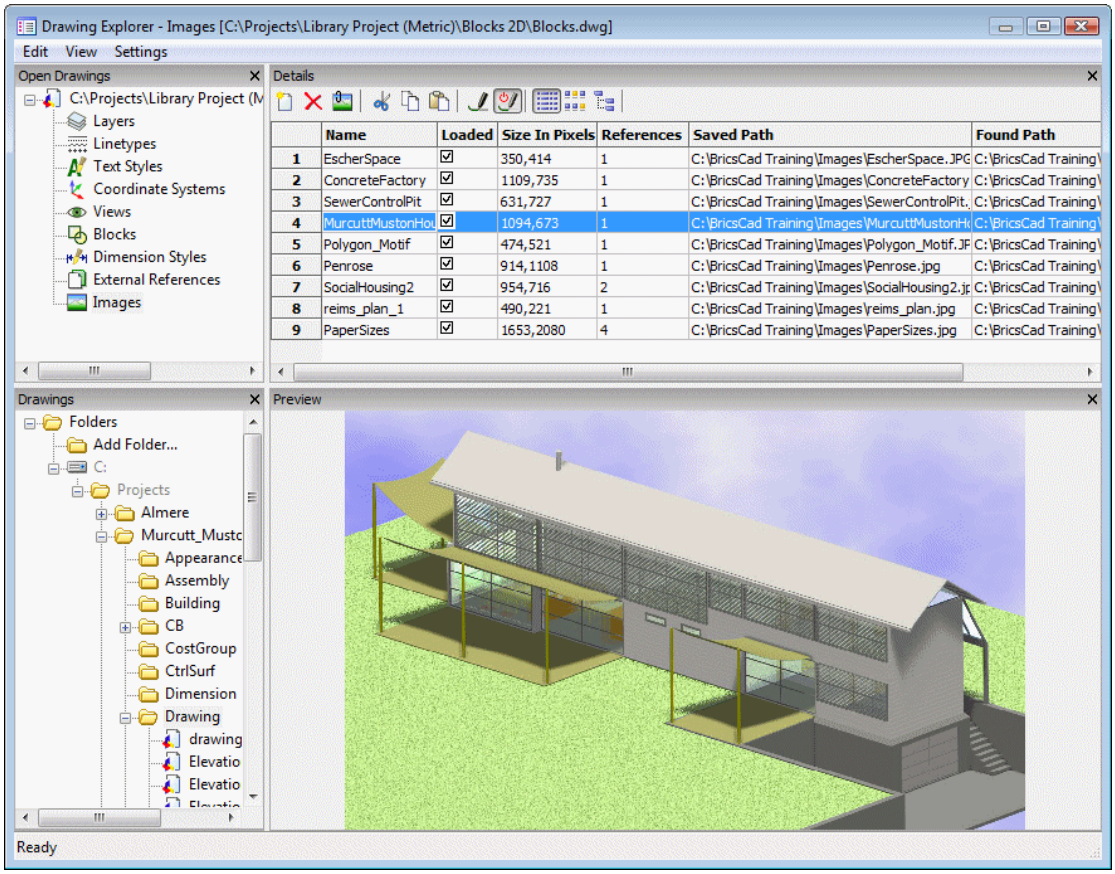
Images Explorer display options

You can choose between *Detail View* (list), *Icon View* and *Tree View* to see the images in the current drawing.

Open the Detail View of the Images Explorer

Click the Detail View button () in the *Drawing Explorer* toolbar. The Detail View button is now pressed (), indicating the image details are displayed.
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Detail View of the images in the current drawing

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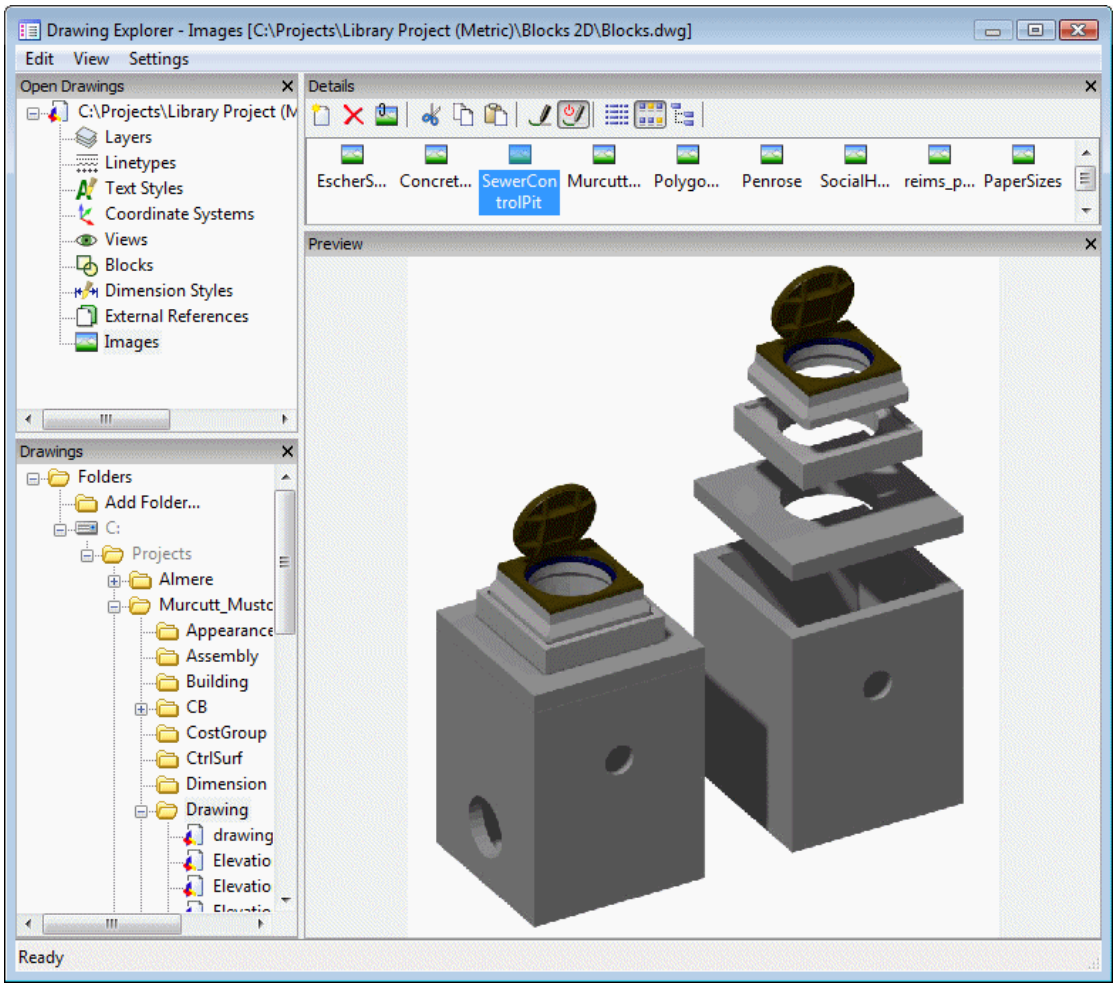
Open the Icon View of the Images Explorer

Click the Icon View button () in the *Drawing Explorer* toolbar. The Icon View button is now pressed (), indicating the image icons are displayed.
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Icon View of the images in the current drawing

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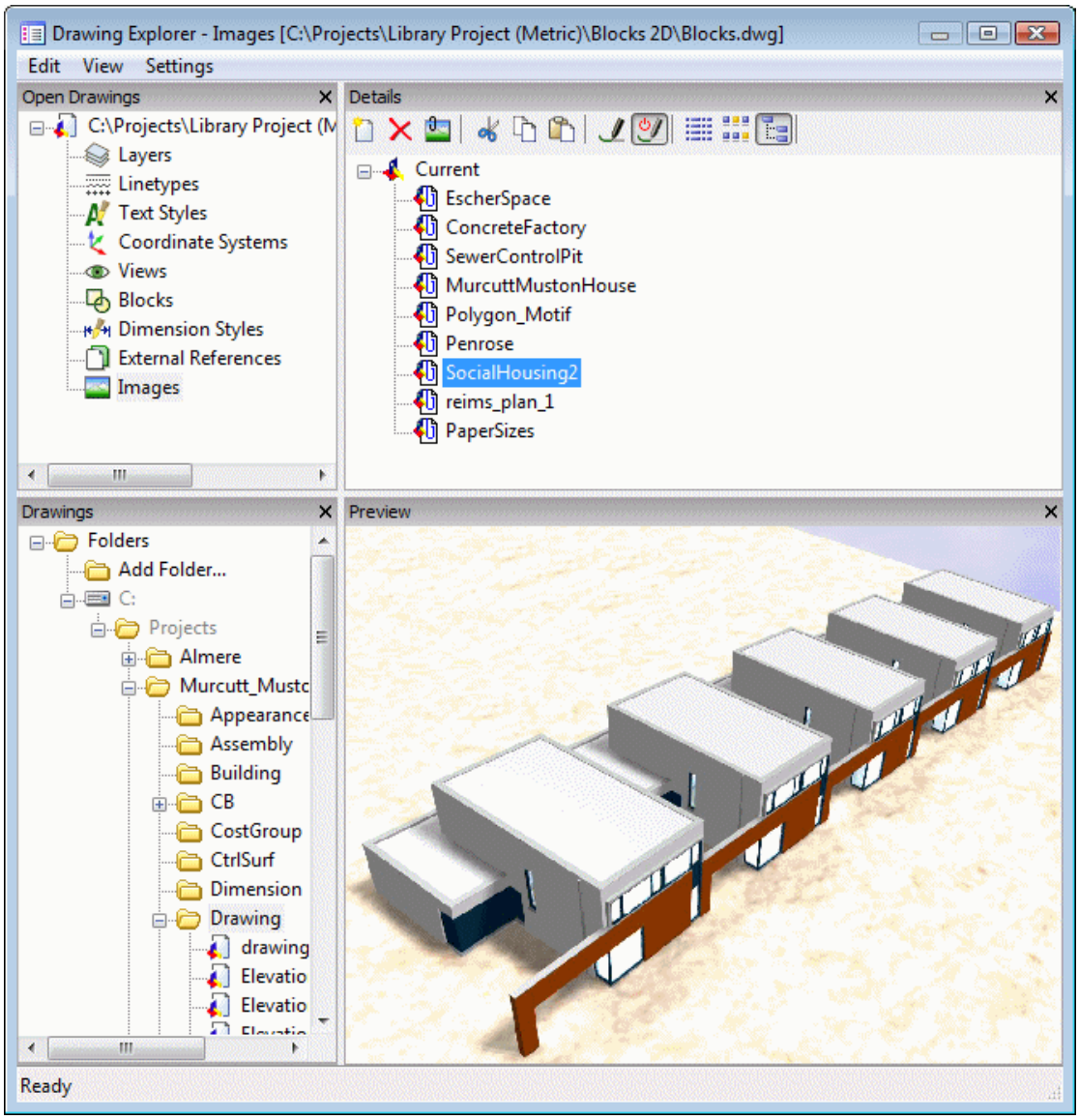
Open the Tree View of the Images Explorer

Click the Tree View button ( ) in the *Drawing Explorer* toolbar. The Tree View button is now pressed ( ), indicating the image tree is displayed.
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Tree View of the images in the current drawing

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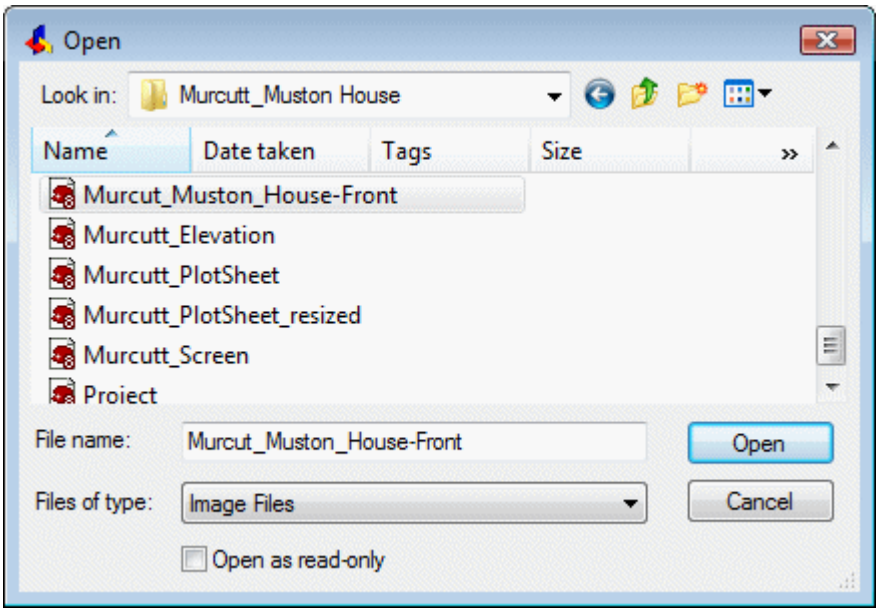
Placing images in a drawing

As different form Bricscad V7, in Bricscad V8 you need to attach an image first, then insert it in the drawing. As a result, an image can be attached to a drawing, while it is not placed in the drawing yet. If you delete an image in the drawing, it will still be attached and can be inserted again.

Attach images

1. Launch the *Image Explorer*.
2. Click the New tool ( ) in the *Drawing Explorer - Images* toolbar.
   The *Open* dialog window displays.

3. In the *Open* dialog window browse to the image.



4. Do one of the following:

- Click the Open button.
- Double click the image.

The image is attached to the current drawing.
A preview of the image shows in *Preview* sub-window of the *Images Explorer*.

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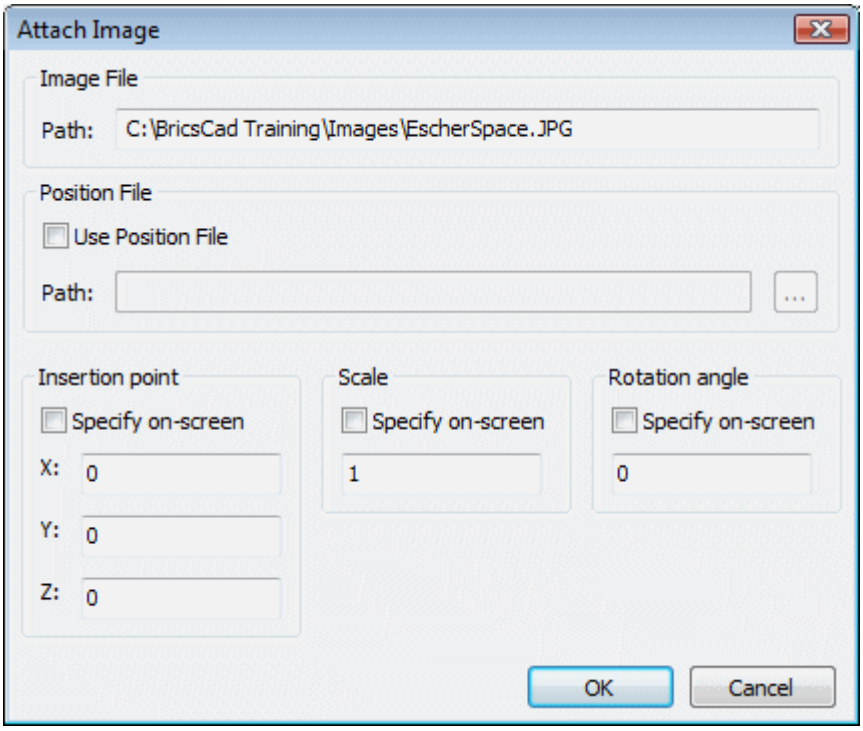
Insert images

1. Launch the *Image Explorer*.
2. Select the image.
3. Do one of the following:

- Click the Insert tool ( ) in the *Drawing Explorer - Images* toolbar.
- Right click and choose Insert in the context menu.

The *Attach Image* dialog opens.



4. On the *Attach Image* dialog:
   - (option) Select a *Position File*.
   - Specify an *insertion point* or select the *Specify on-screen* check box.
   - Specify *scale* or select the *Specify on-screen* check box.
   - Specify *rotation* angle or select the *Specify on-screen* check box.
5. Click the *OK* button.
6. If any of the *Specify on-screen* options is selected, the image is inserted at the desired insertion point, scale and rotation angle;

else

Specify the insertion point, scale and/or rotation angle on-screen.

NOTE The Attach Raster Image tool in the *Images* toolbar simultaneously attaches and inserts the image.

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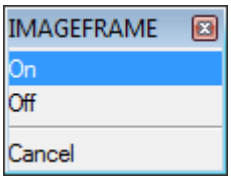
Load / Unload images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily remove it. Unloaded images are still inserted in the drawing, but they no longer display. If the Imageframe setting is on, the frame still displays though.

Modify the Image Frame setting

1. Click the Image Frame tool ( ) in the Images toolbar.

2. Choose *On* or *Off* on the *Imageframe* option box.

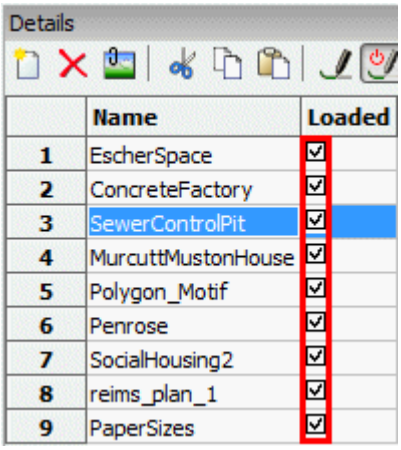


The display of the image frames changes accordingly.

go to top

Load / Unload a single image

1. Launch the *Image Explorer*.
2. For the image you want to load or unload, click the check box in the *Loaded* column.



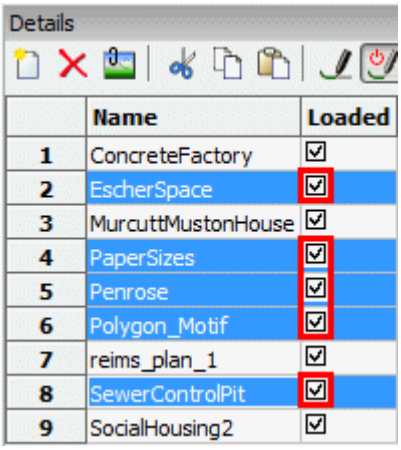
3. If the Regen On/Off button ( ) is not pressed, click the Regen tool ( ) in the *Images Explorer* toolbar.

NOTE It is not necessary to select the image first in step 2.

go to top

Load / Unload multiple images

1. Launch the *Image Explorer*.
2. Select the images you want to load or unload.
3. For one of the selected images, click the check box in the *Loaded* column.



All selected images will be loaded or unloaded simultaneously.

4. If the Regen On/Off button ( ) is not pressed, click the Regen tool ( ) in the *Images Explorer* toolbar.

NOTE Press and hold the Ctrl key to select multiple images.

go to top

Removing images

Since an image is first attached, then inserted in the drawing, you can delete an image, without detaching it.

Detach images

1. Launch the *Image Explorer*.
2. Select the image(s) that you want to detach.
3. Do one of the following

- Click the Delete tool ( ) in the *Images Explorer* toolbar.
- Right click and select Delete in the context menu.

NOTE Press and hold the Ctrl key to select multiple images.

go to top

Bricscad Properties Bar

Bricscad Properties Bar

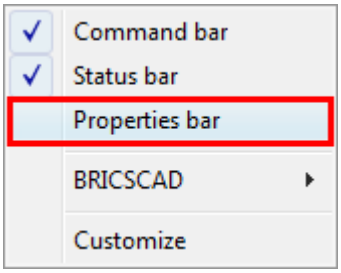
In the Bricscad Properties Bar you can:

- Set the current properties: *color*, *layer*, *linetype*, *linetype scale* and *lineweight*
- See the properties of a single entity
- Edit the properties of a single entity
- Edit endpoints of lines and and vertices of polylines graphically
- Edit the shared properties of a selection set

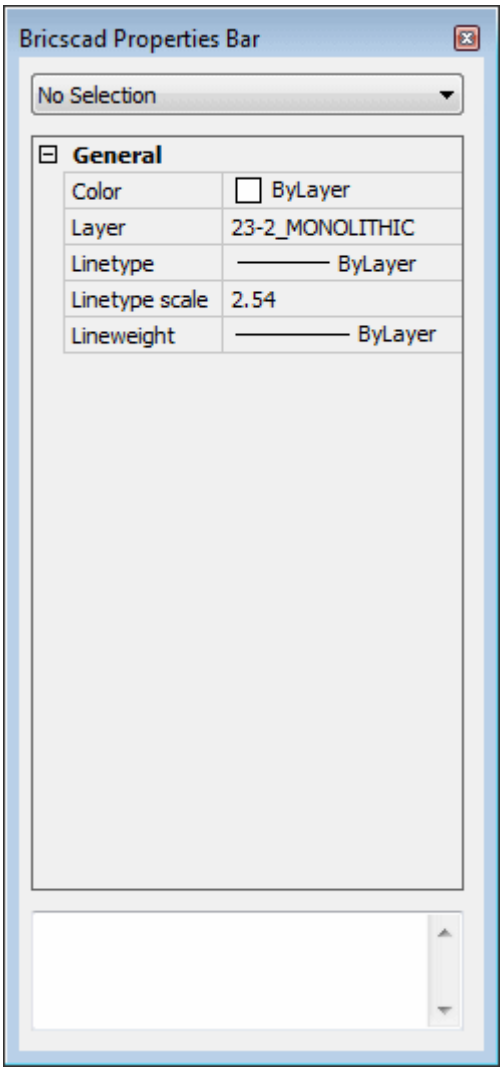
Open the Properties Bar

Do one of the following.

- Right click when the cursor is on a toolbar.
  A context menu displays.
  The marked items in the context menu are currently open.
  Select *Properties Bar* in the context menu.



- Double click an entity. The *Bricscad Properties Bar* opens, showing the properties of the selected entity.
- Click the Properties tool button ( ) on the *Modify* toolbar.
- Choose *Properties* in the *Modify* menu.
- Type *properties* in the command window, then press Enter.



NOTE The *Bricscad Properties Bar* can be either floating or docked.

To dock the *Bricscad Properties Bar*, drag it by its title bar to either the left or right hand side of the Bricscad application window.

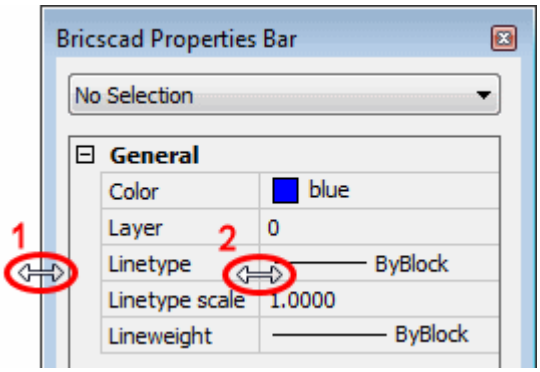
go to top

Adjust the size of the Bricscad Properties Bar

1. Move the cursor over one of the edges of the *Bricscad Properties Bar* (1).

   The cursor turns into a double-headed arrow.

2. Press and hold the left mouse button to drag the edge of the *Bricscad Properties Bar*.



3. Move the cursor over the boundary between the *Setting Names* and the *Setting Fields* columns (2).

The cursor turns into a double-headed arrow.

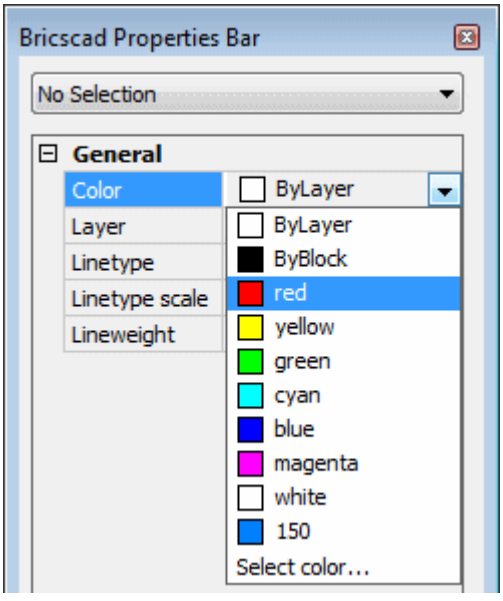
4. Press and hold the left mouse button to drag the boundary.

NOTE When docked, only the left (or right) edge of the *Bricscad Properties Bar* is adjustable.

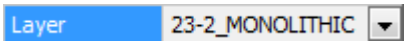
Set the current properties in the Bricscad Properties Bar

1. (option) Click on *Color,*

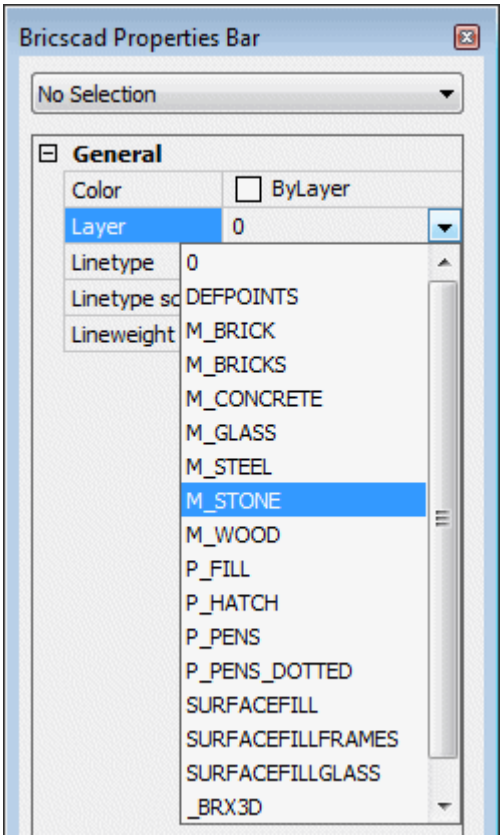
then click the down arrow button to select a color.



2. (option) Click on *Layer,*

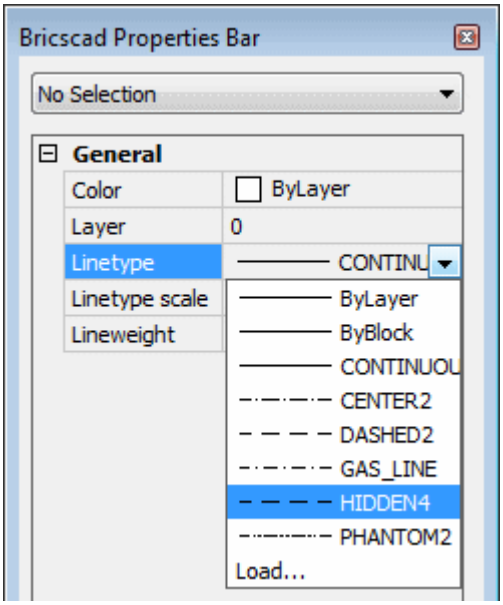


then click the down arrow to select a layer.

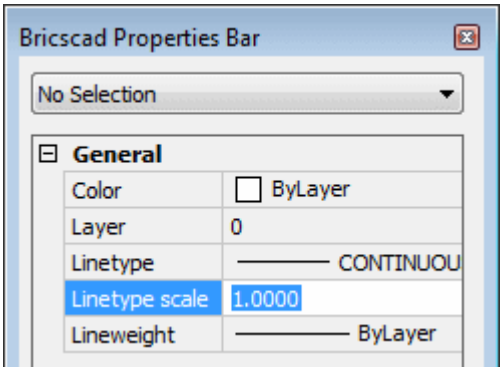


3. (option) Click on *Linetype*.

then click on the down arrow to select a linetype.



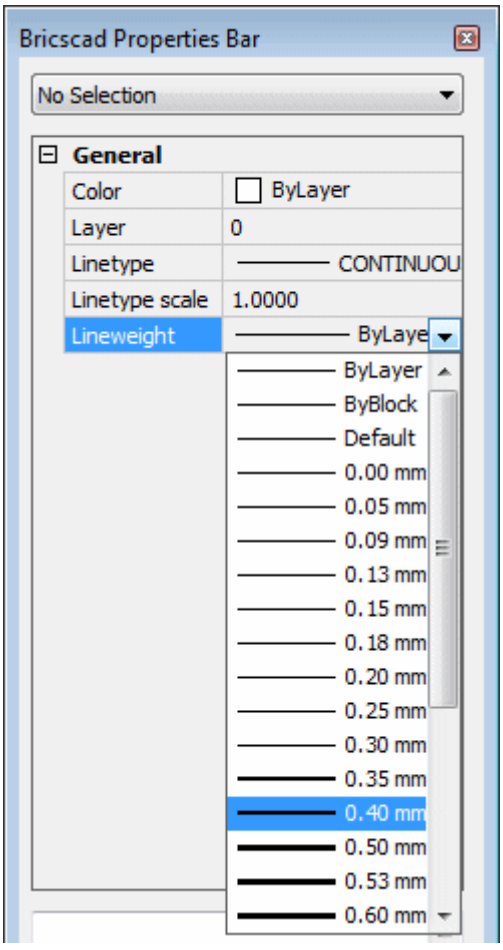
4. (option) Click on *Linetype scale*,



then type the new value in the *Linetype scale* field.

5. (option) Click on Lineweight,

then click on the down arrow to select a lineweight.

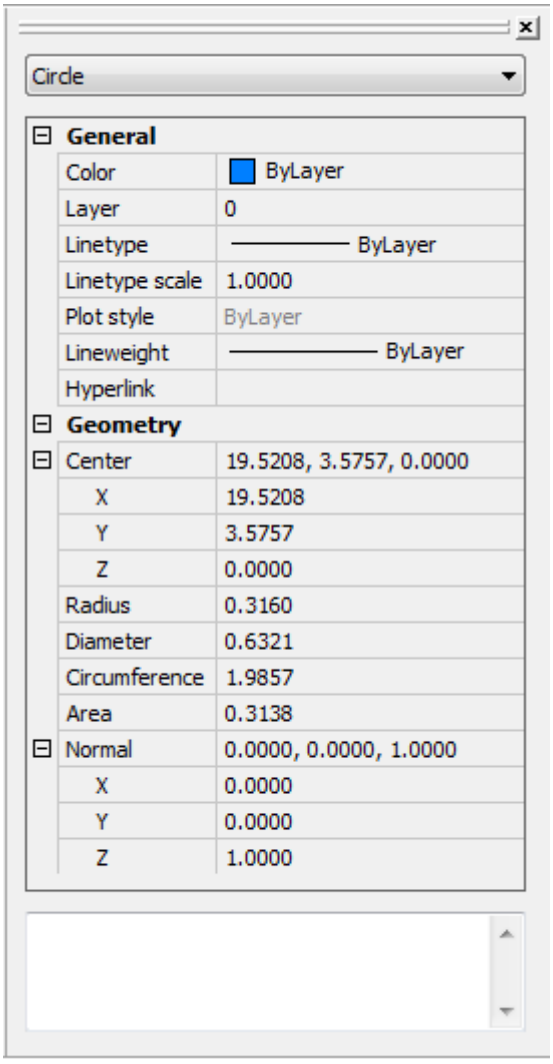


go to top

See the properties of a single entity

1. Select the entity.

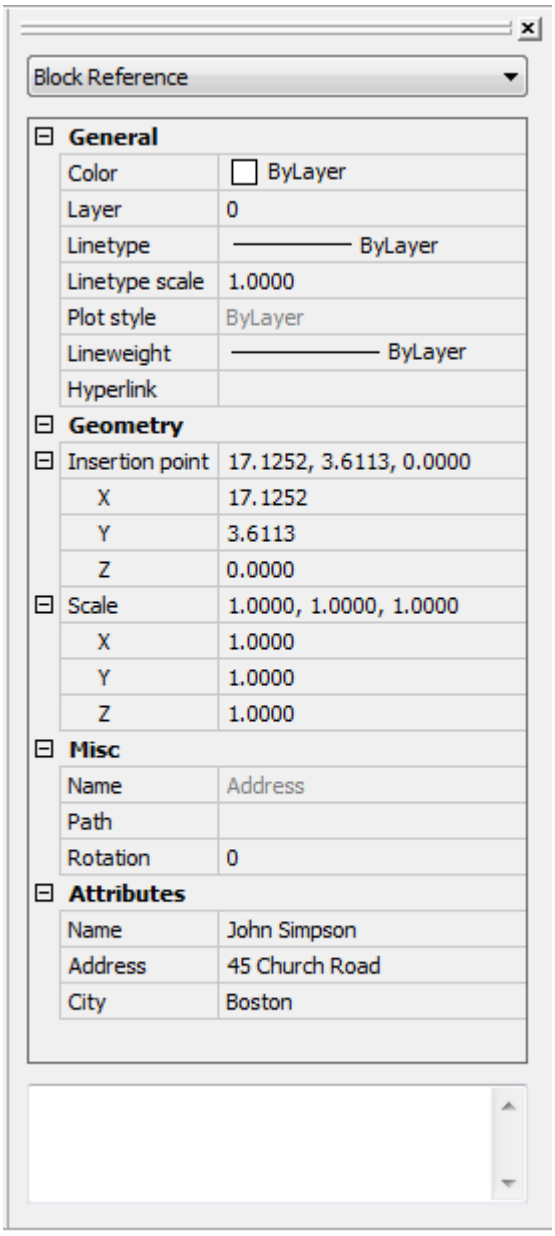
   The entity's properties display in the *Bricscad Properties Bar*.



Properties of a circle

2. (option) Press the *Escape* key to deselect the previous entity and select another entity.

The newly selected entity's properties display in the *Bricscad Properties Bar*.



Properties of a block

3. Press the *Escape* key to stop.

go to top

Edit the properties of a single entity

1. Select the entity.
   The entity's properties display in the *Bricscad Properties Bar*.
2. Click the property you want to modify.
   The settings field of the selected property is activated.

3. Type a new value in the settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
   The entity is updated.
4. (option) repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

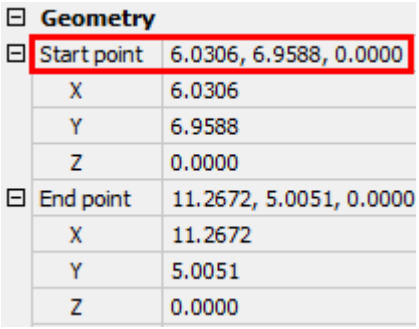
NOTES

- Properties of which the value displays in grey cannot be modified.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

go to top

Edit endpoints of a line graphically

1. Select the line.
2. On the *Bricscad Properties Bar*, under *Geometry*, select either *Start point* or *End point*.



Geometry	
Start point	6.0306, 6.9588, 0.0000
X	6.0306
Y	6.9588
Z	0.0000
End point	11.2672, 5.0051, 0.0000
X	11.2672
Y	5.0051
Z	0.0000

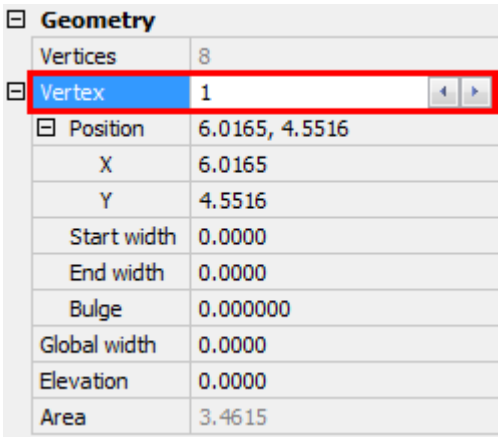
   An X indicates the point being edited.
3. Click the *Define* button ( ).
4. In the drawing, specify the point graphically.
   The XYZ coordinates of the point are adjusted accordingly.
5. Press the *Escape* key to stop.

NOTE This procedure also applies to the edit the insertion point of blocks.

Edit vertices of polyline graphically

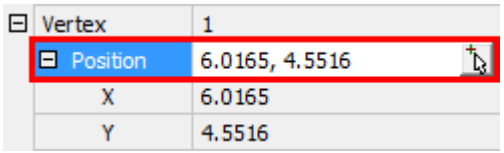
1. Select the polyline..
2. On the *Bricscad Properties Bar* under *Geometry*, select *Vertex*.

3. Click the *Next/Previous* arrow buttons to select a vertex.



An X indicates the vertex being edited.

4. Under *Vertex* select *Position*.



5. Click the *Define* button ().
6. In the drawing, specify the point graphically.
   The XYZ coordinates of the point are adjusted accordingly.
7. Press the *Escape* key to stop.

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Edit the shared properties of a selection set

1. Select the entities.
   The shared properties display in the *Bricscad Properties Bar*.
2. Click the property you want to modify.
   The settings field of the selected property is activated.
3. Type a new value in settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
   All selected entities are updated simultaneously.
4. (option) Repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

NOTES

- *Varies* displays for shared properties which are defined differently. If you edit such property, all entities in the selection set will be equally defined for this property.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

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Settings Dialog

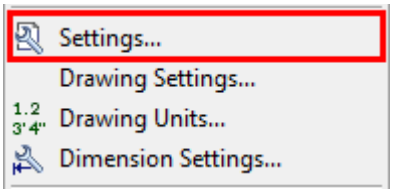
Settings Dialog

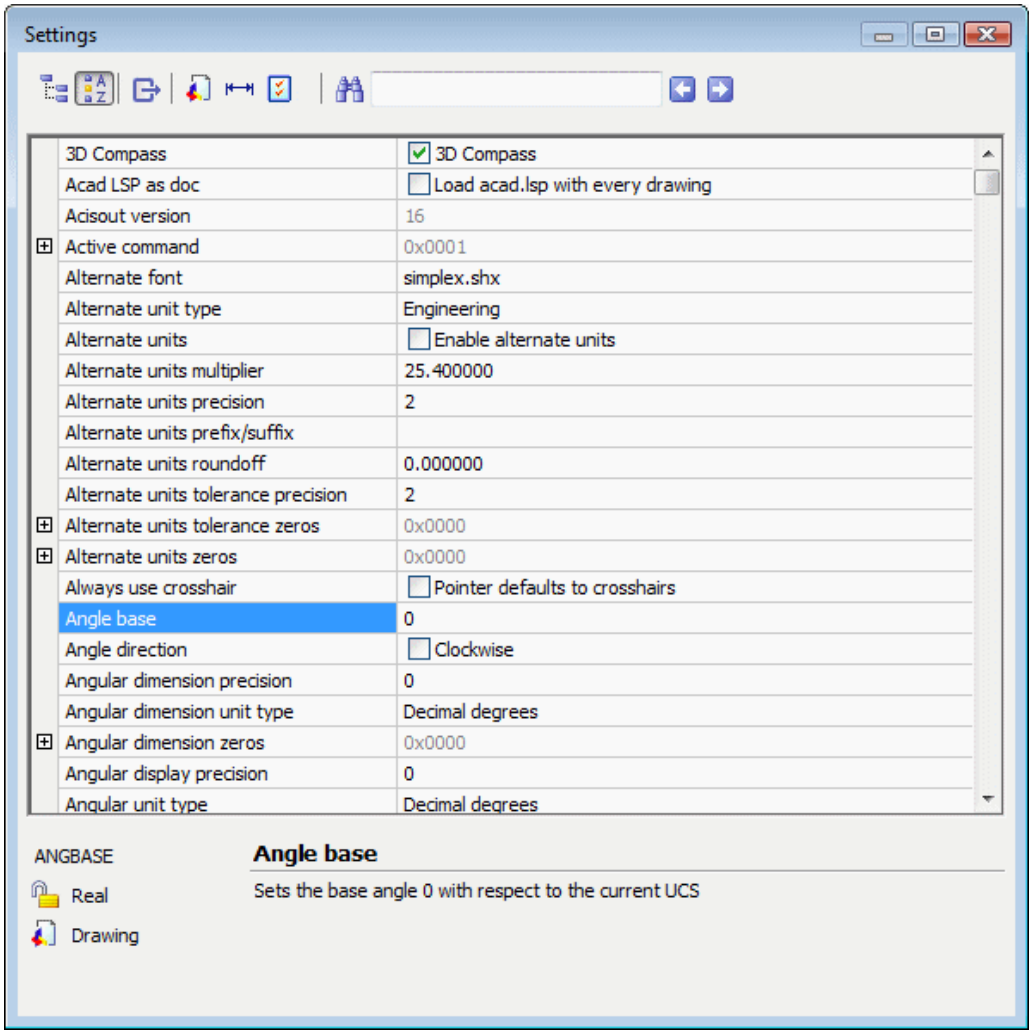
In the *Settings Dialog* you can:

- Check the current value of all settings.
- Edit settings.
- Find settings.
- Export the settings in a text file

Open the Settings dialog

1. Choose *Settings* in the *Settings* menu.





2. (option) Choose a display mode by clicking the corresponding button in the *Settings* toolbar.

Categorized ( )
Lists the settings by category: *Drawing*, *Dimensions* and *Program Options*.

Alphabetical ( )
Lists the settings alphabetically.

3. (option) Choose a category:

Drawing ( )
Opens the *Categorized* view mode, with the *Drawing* settings tree expanded.

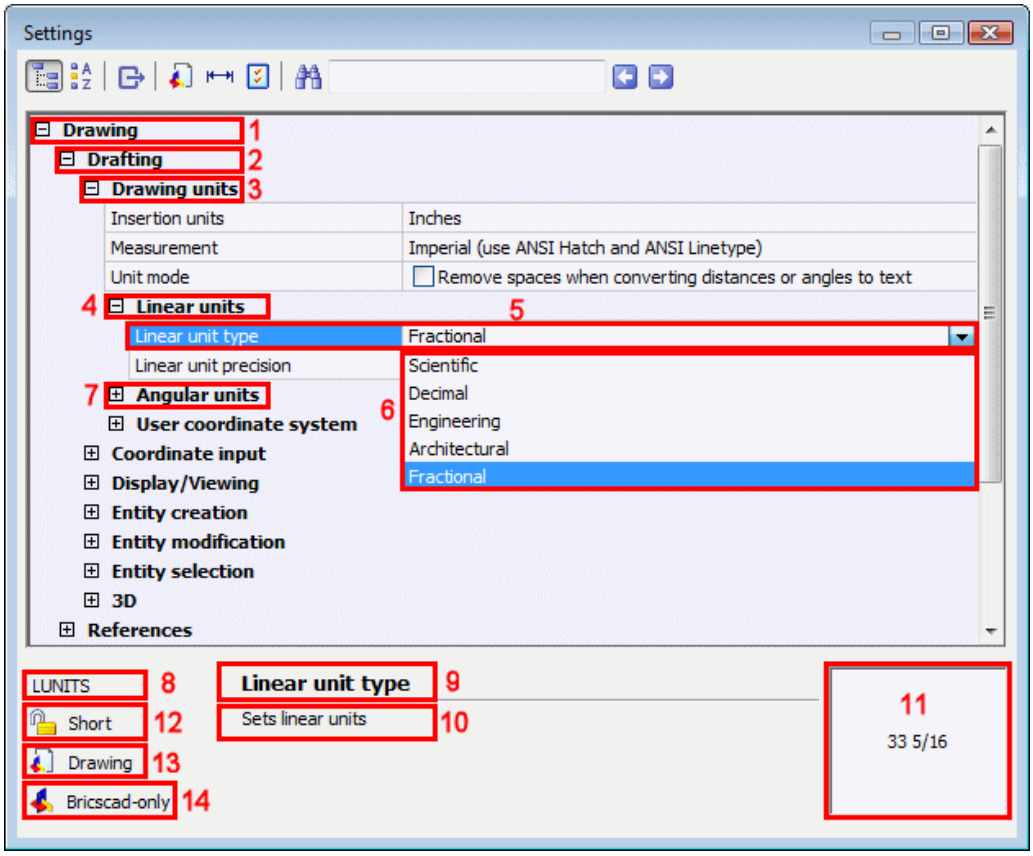
Dimensions ( )
Opens the *Categorized* view mode, with the *Dimension setting*s tree expanded.

Program Options ( )
Opens the *Categorized* view mode, with the *Program Options setting*s tree expanded.

go to top

Edit settings

1. Open the Settings Dialog.
2. Select a setting.



1	Class: Drawing, Dimensions or Program Options
2	Category (expanded)
3	Sub-category (expanded)
4	Group (expanded)
5	Setting and its current value
6	Options list
7	Group (collapsed)
8	Command (to define the setting in the command bar)
9	Settings Name
10	Settings Description
11	Preview
12	Settings Type (= hard coded; = user-defined)
13	Where is the setting saved? (= drawing; = registry; = not saved)
14	(Bricscad-only) Indicates a setting is available in Bricscad only.

4. Click the settings field to edit the setting.
5. (option) Repeat steps 3 and 4 to edit more settings.
6. Close the *Settings* window.

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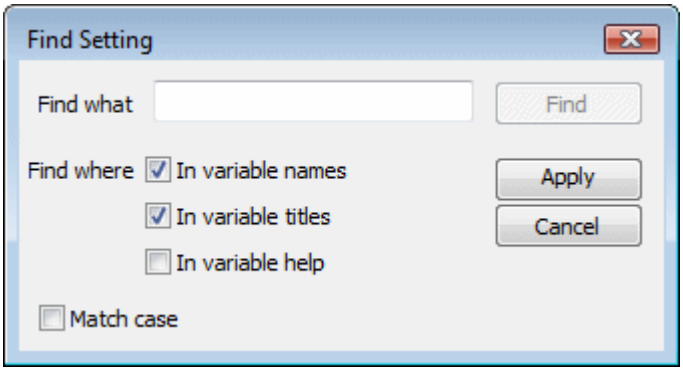
Find Settings

1. Open the Settings Dialog.
2. (option) Click the *Find Options* button ( ) to open the open the *Find Setting* dialog where you can set the search options.



3. Type the search string in the *Search* field.

The first setting matching the content of the search field highlights.

4. (option) Click the *next* () or *previous* () button to browse through the matching settings.

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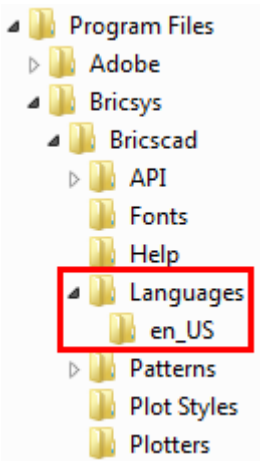
Export Settings

1. Click the Export tool () on the *Settings* toolbar.
   The *Export Settings* window opens.
2. Type a name in the *File name* field.
3. (option) Select a folder.
   By default the export file is saved in the *Languages/en_US* folder of the Bricscad installation folder
   e.g. C:\Program Files\Bricsys\Bricscad\Languages\en_US.



4. Click the Save button on the *Export Settings* window.
   All Settings are exported in a CSV (**C**omma **S**eparated **V**alues) text file, which can be opened in Microsoft® Excel®.

	A	B	C	D	E	F	G	H
1	Name	SaveMode	SaveType	ResType	DefaultValue	CurrentValue	Status	Title
2								
3	AFLAGS	not	int	RTSHORT	0	0		Default attribute modes
4	ANGBASE	drw	real	RTREAL	0	0		Angle base
5	ANGDIR	drw	bool	RTSHORT	0	0		Angle direction
6	AREA	not	real	RTREAL		0	readOnly	Area
7	ATTDIA	reg	bool	RTSHORT		0		Attribute dialog
8	ATTMODE	drw	int	RTSHORT	1	1		Attribute display mode
9	ATTREQ	reg	bool	RTSHORT	1	1		Insertion default settings
10	AUNITS	drw	int	RTSHORT	0	1		Angular unit type

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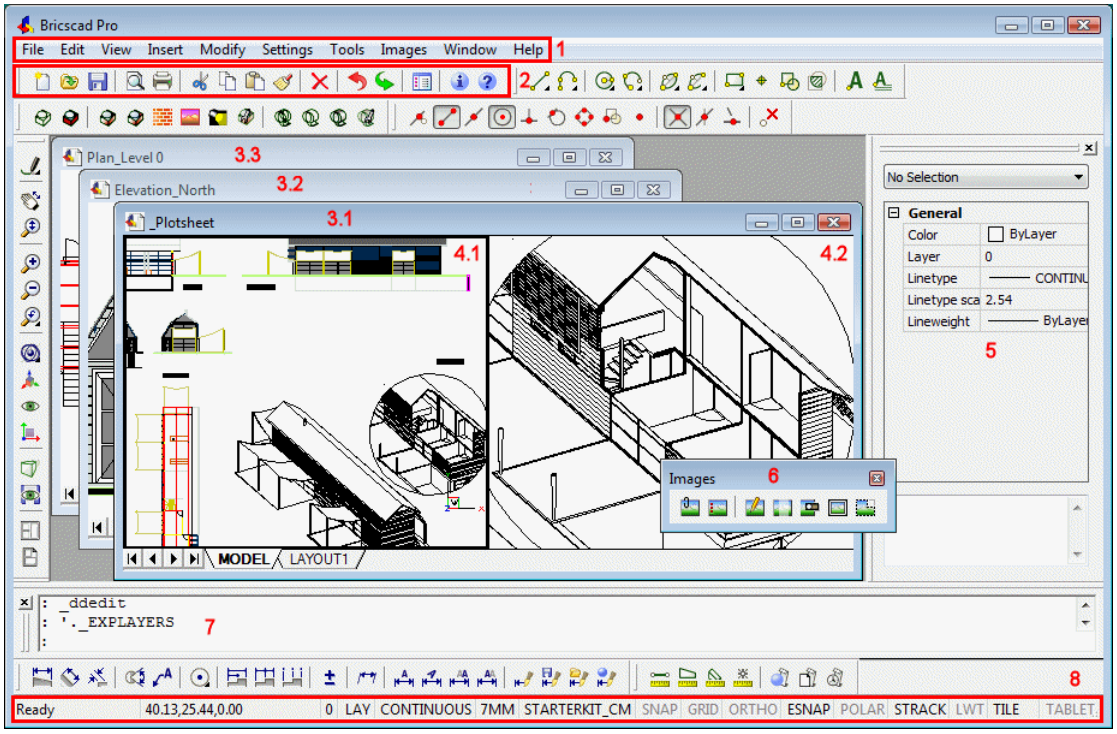
The Bricscad Application Window

The layout of the Bricscad application window can be fully customized.
You can:

- open / close the command window
- open / close the status bar
- modify drawing settings in the status bar
- display / hide scroll bars



Components of the Bricscad application window:

1. Menu Bar
2. Toolbar (docked)
3. Drawing windows
4. Drawing viewports
5. Bricscad Properties Bar
6. Toolbar (floating)
7. Command window
8. Status Bar

Open / Close the command bar

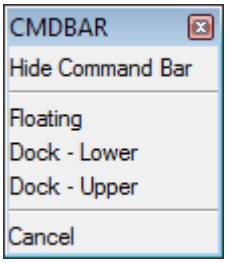
1. (option) Do one of the following:

- Choose *Command Bar* in the *View* Menu.
- Move the cursor to a docked toolbar, then right click and choose *Command Bar* in the context menu.

The *Command Window* closes if it was open and vice versa.

2. (option) Type cmdbar.

The *CMDBAR* context menu opens.



3. Choose one of the options on the *Cmdbar* context menu.

NOTES

- You can modify the height of the docked command window by dragging its top edge, when docked *Lower* or bottom edge, when docked *Upper*.
- The size of a floating command bar can be adjusted by dragging one of its edges.
- When the command bar is closed, command options and keyboard entries display in the status bar.

Open / Close the status bar

Do one of the following:

- Choose Status Bar in the *View* Menu.
- Press the *F10* function key on the keyboard.

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

Working with the status bar

The *Status Bar* fields are:

1. Status:

- displays the status of the software.
- when the cursor is in a menu or on a toolbar: gives a brief description of the tool or menu item.

- when the command window is closed: displays the tool options and keyboard entry.

2. Coordinates: displays the coordinates of the current cursor position (depending on the setting of the COORDS variable).

3. Layer: displays the name of the current layer.

- double click to open the *Layer Explorer*.
- right click to select the current layer

4. Color: displays the current color.

- double click to open the *Select Color* dialog window.
- right click to select one of the basic colors.

5. Linetype: displays the name of the current linetype.

- double click to open the *Linetype Explorer*.
- right click to select the current linetype

6. Text Style: displays the name of the current text style.

- double click to open the *Styles Explorer*.
- right click to select the current text style.

7. Dimension Style: displays the name of the current dimension style

- double click to op en the *Dimensions Settings* dialog window.
- right click to select the current dimension style.

8. Snap:

- double click to toggle *Snap* on (SNAP) /off (SNAP).
- right click, then choose *Settings* to get access to the *Snap and Grid* settings.

9. Grid: double click to toggle the display of *grid points* on (GRID) / off (GRID) in the current viewport.

10. Orthogonal Mode: double click to toggle *Orthogonal mode* on (ORTHO) /off (ORTHO). Turning *Orthogonal mode* on automatically disables *Polar Tracking*.

11. Entity Snaps:

- double click to toggle *Entity Snaps* on (ESNAP) / off (ESNAP).
- right click, then choose *Settings* to get access to the *Entity Snaps* settings.

12. Polar Tracking (AutoSnap):

- double click to toggle *Polar Tracking* on (POLAR) / off (POLAR).
  Turning *Polar Tracking* on automatically disables *Orthogonal mode*.
- right click, then choose *Settings* to get access to the *Polar Tracking* settings.

13. Snap Tracking (AutoSnap):

- double click to toggle *Snap Tracking* on (STRACK) /off (STRACK).
- right click, then choose *Settings* to get access to the *Snap Tracking* settings.

14. Line Weight Dispaly:

- double clik to toggle the display of *Line Weights On* (LWT) or *Off* (LWT).
- right click, then choose *On* or *Off* to control the display of *Line Weights*.

15. Current Workspace.

- *Tile :* Model space with tiled viewports.
- *M:Layout :* Model space with floating viewports
- *P:Layout* : Paper space

16. Tablet: Initializes the use of a drawing tablet.

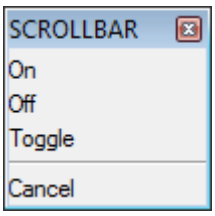
NOTE *Orthogonal mode* (field 10) is switched off if *Polar Tracking* (field 14) is on and vice versa.

Display or hide scroll bars

1. (option) To toggle scroll bars on / off: choose Scroll Bars in the *View* menu.

2. (option) Type *scrollbar* in the command bar, then do one of the following:

- type *off* in the command bar or choose *Off* in the context menu.
- type *on* in the command bar or choose *On* in the context menu.
- type *T* in the command bar or choose *Toggle* in the context menu.



Toolbars

Toolbars

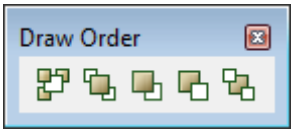
The following toolbars are available in Bricscad.

Standard

Draw 2D

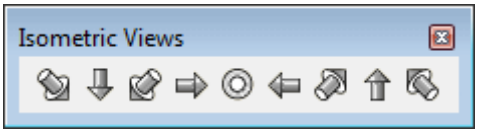
Modify

Draw Order

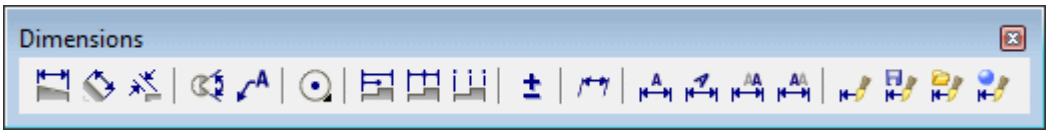


View

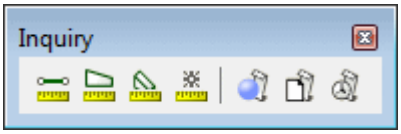
Isometric Views



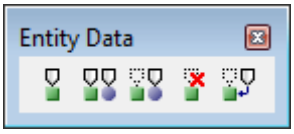
Dimensions



Inquiry



Entity Data



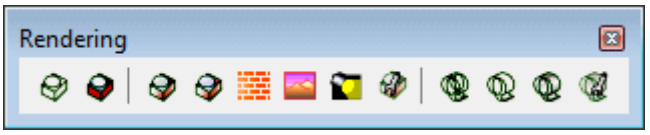
Entity Snaps

Draw 3D

Solid Editing

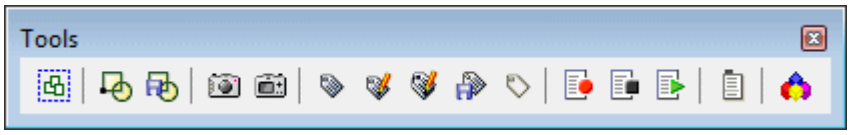
Solids

Rendering

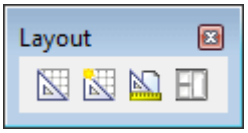


Settings

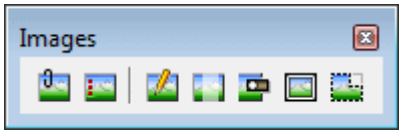
Tools



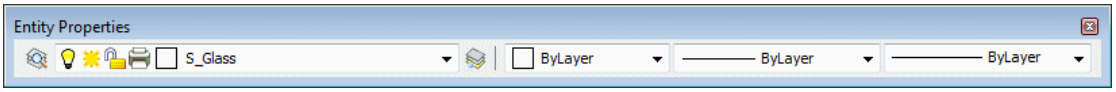
Layout



Images

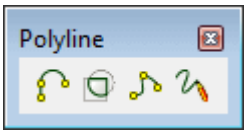


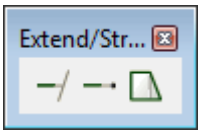
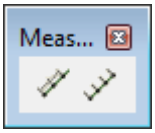
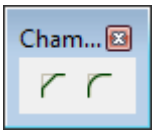
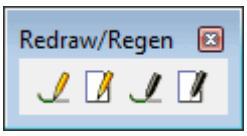
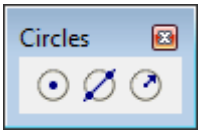
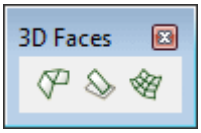
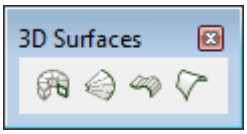
Entity Properties



Toolbar Flyouts

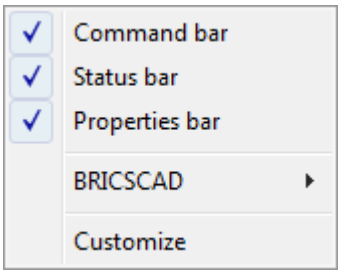
On some tool buttons a small black arrow indicates a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar. Each of the flyouts can also be opened as a separate toolbar.

Parent	**Flyout**	
Draw 2D	Line	
	Polyline	
	Polygon	

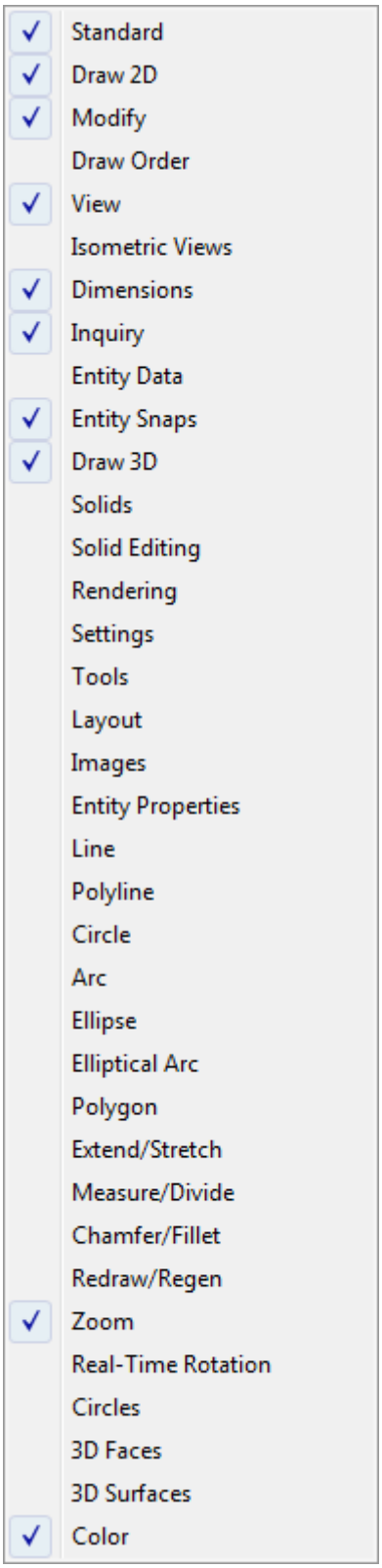
Modify	Extend/Stretch	
	Measure/Divide	
	Chamfer/Fillet	
View	Redraw/Regen	
	Zoom	
	Real-Time Rotate	
Dimensions	Circles	
Draw 3D	3D Faces	
	3D Surfaces	
Settings	Color	

Opening a toolbar

1. Place the cursor on a open toolbar, then right click.
   A context menu displays.



2. Choose *BRICSCAD* on the context menu.
   A list of all available toolbars displays. Toolbars that are already open are checked.



2. Select the toolbar on the context menu.

The toolbar opens.

3. (option) Dock the toolbar by dragging the toolbar by its title bar to one of the edges of the Bricscad application window.

NOTES

- If you select a toolbar that was already open, it will be closed.
- To prevent a toolbar from docking, press and hold the Ctrl key while dragging the toolbar.
- Drag a toolbar by its left (or top) edge () to undock.

Closing a toolbar

1. Place the cursor on a open toolbar, then right click.
   A context menu displays.
2. Choose *BRICSCAD* on the context menu.
3. On the toolbar list click the toolbar you want to close.

NOTE You can close a floating toolbar also by clicking its *Close* button ().

Customizing toolbars

Not yet implemented.

Composing a user-defined toolbar

Not yet implemented.

Defining your preferences

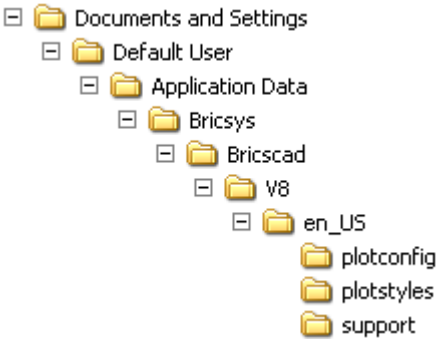
Overview

In Bricscad all preferences of each user are stored in specific folders, which are called the *Local root* folder and the *Roamable root* folder.

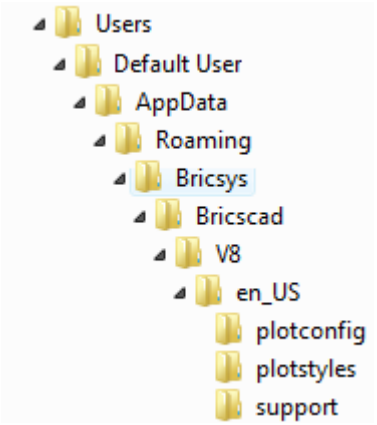
The *Local root* folder contains the *Template* folder, where the drawing templates are saved.

The *Roamable root* folder contains three subfolders:

- *plotconfig*: where the plotter configuration files (*.pc3) are saved
- *plotstyles*: where the plot style table files (*.stb) and plot color table files (*.ctb) are saved
- *support*: where the hatch pattern files (*.pat), line style files (*.lin), alias files (*.pgp), unit files (*.unt), custom user interface files (*.cui), etc. are saved.



Roamable root folder in Windows XP
C:\Documents and Settings\<user name>\Application Data\Bricsys\Bricscad\V8\en_US



Roamable root folder in Windows Vista
C:\Users\<user name>\AppData\Roaming\\Bricsys\Bricscad\V8\en_US

When you install an update of the software, the first time you launch Bricscad after the update, the content of the *User Data Cache* folder (C:\Program Files\Bricsys\Bricscad\UserDataCache) is compared with the content of the *Local root* folder and the *Roamable root* folder of the current user. If you have customized the files in these folders and if one or more of the corresponding files in the *User Data Cache* folder are more recent, the *Bricscad user file manager* is launched asking you whether to keep your files or to overwrite your files with the updated files.

Using the Bricscad user file manager

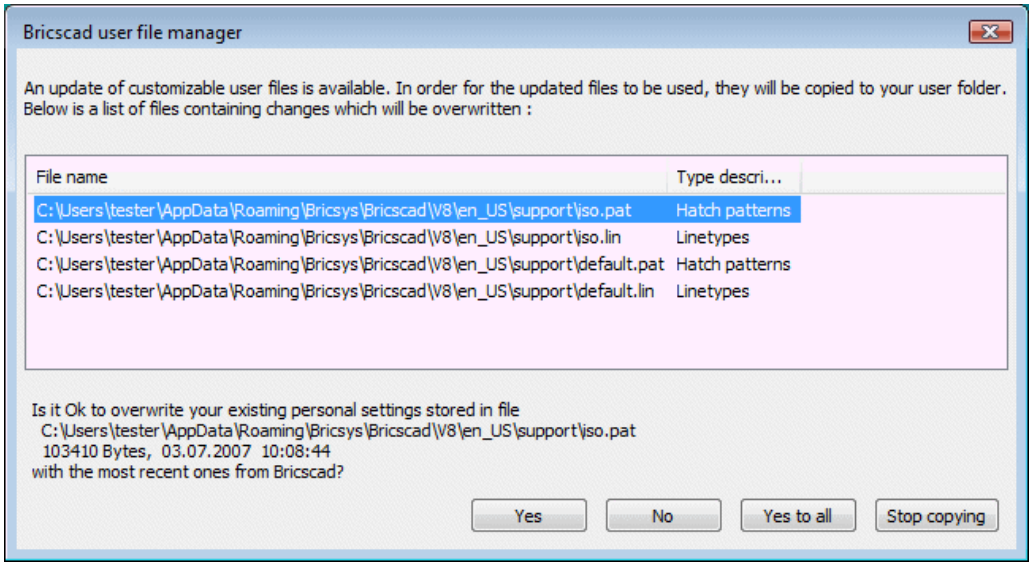
1. Do one of the following:

- Select a file in the file list, then click the *Yes* button to overwrite or click the *No* button to keep your file.

  Repeat this procedure for each file in the list.

- Click the *Yes to all* button to replace all your files with the more recent files from Bricscad.

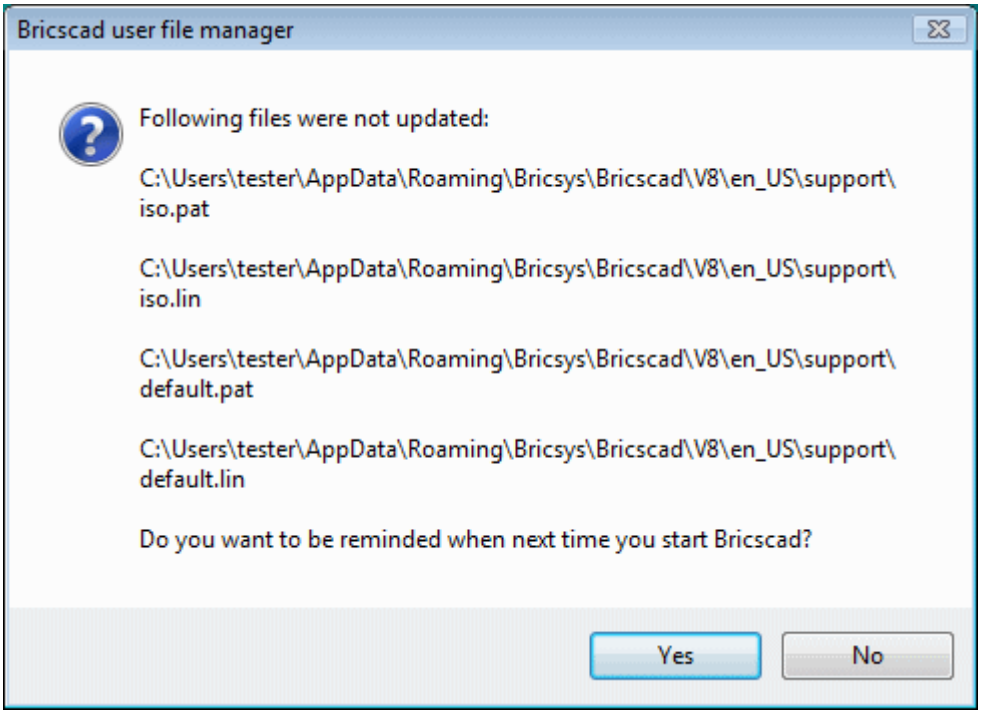
- Click the *Stop Copying* button to keep all your files.



2. If you have decided not to update one or more files, an alert window displays.

Do one of the following:

- Click the *Yes* button if you want to update one of these files later.
  In this case this procedure is restarted the next time you start Bricscad.
- Click the *No* button if you don't want to be reminded.



Drawing Accurately

Overview

In Bricscad the following drawing aids are available:

- Coordinate Input
- Snap and Grid
- Drawing Limits
- Ortho Mode
- Entity Snaps
- Polar Tracking
- Snap Tracking
- User Coordinate Systems
- Direct Distance Entry

Coordinate Input

Coordinate Input

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

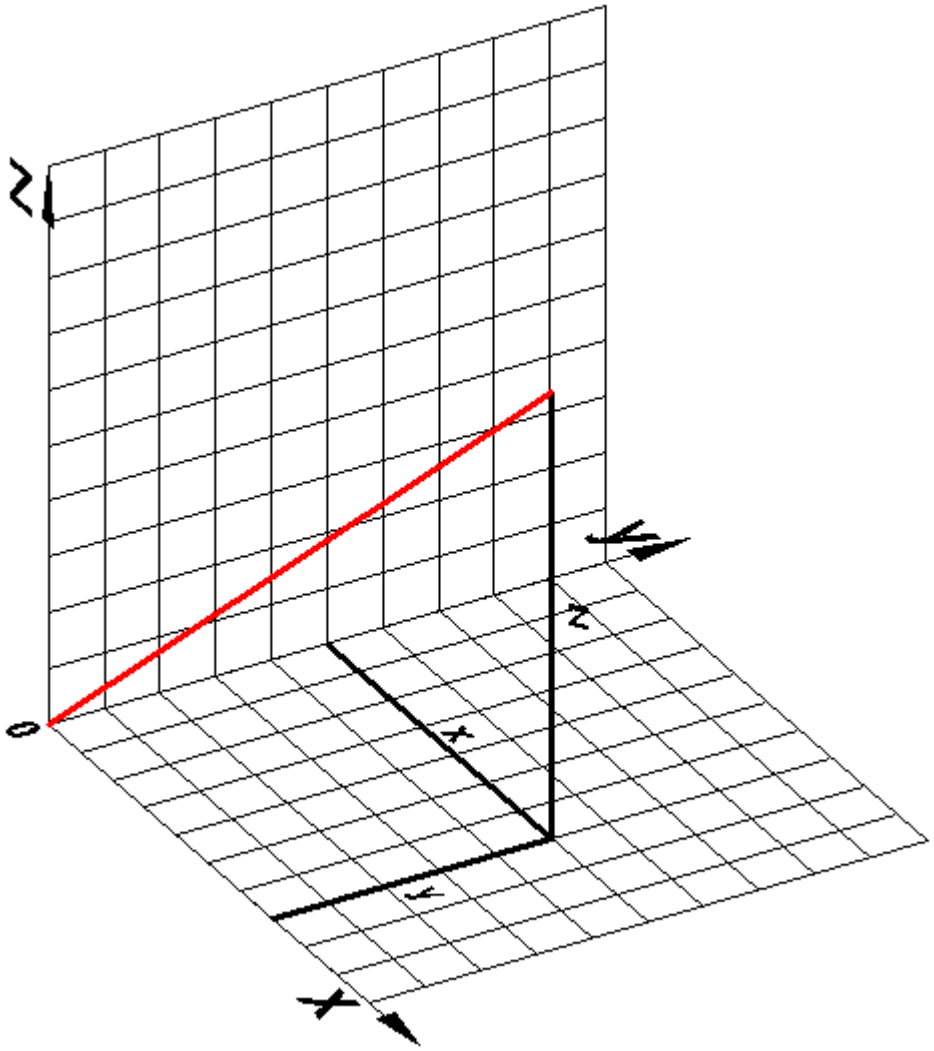
You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates
- Cylindrical coordinates
- Spherical coordinates

Working with Cartesian coordinates

In the Cartesian coordinate system we use three perpendicular axis: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x- and y-axes define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the xy-, xz- and yz- planes. These distances are called the xyz-coordinates of a point.



If you want to enter the absolute Cartesian coordinates of a point, type the x-, y- and z-coordinates separated by commas: 45.5,57.3,60

If you omit the z-coordinate, the point is placed in the xy-plane (Z = 0).

If you place the @ - character in front of the entry, the coordinates are calculated with respect to the previous point: @45.5,57.3,60. This technique is called Relative Cartesian coordinates.

Using relative Cartesian coordinates to draw a rectangle

1. Launch the Rectangle command.
2. Specify the first corner of the rectangle.
3. In the command bar type: @<width>,<height>

- <width> = the width of the rectangle in drawing units, measured along the x-axis
- <height> = the height of the rectangle in drawing units, measured along the y-axis

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Working with cylindrical coordinates

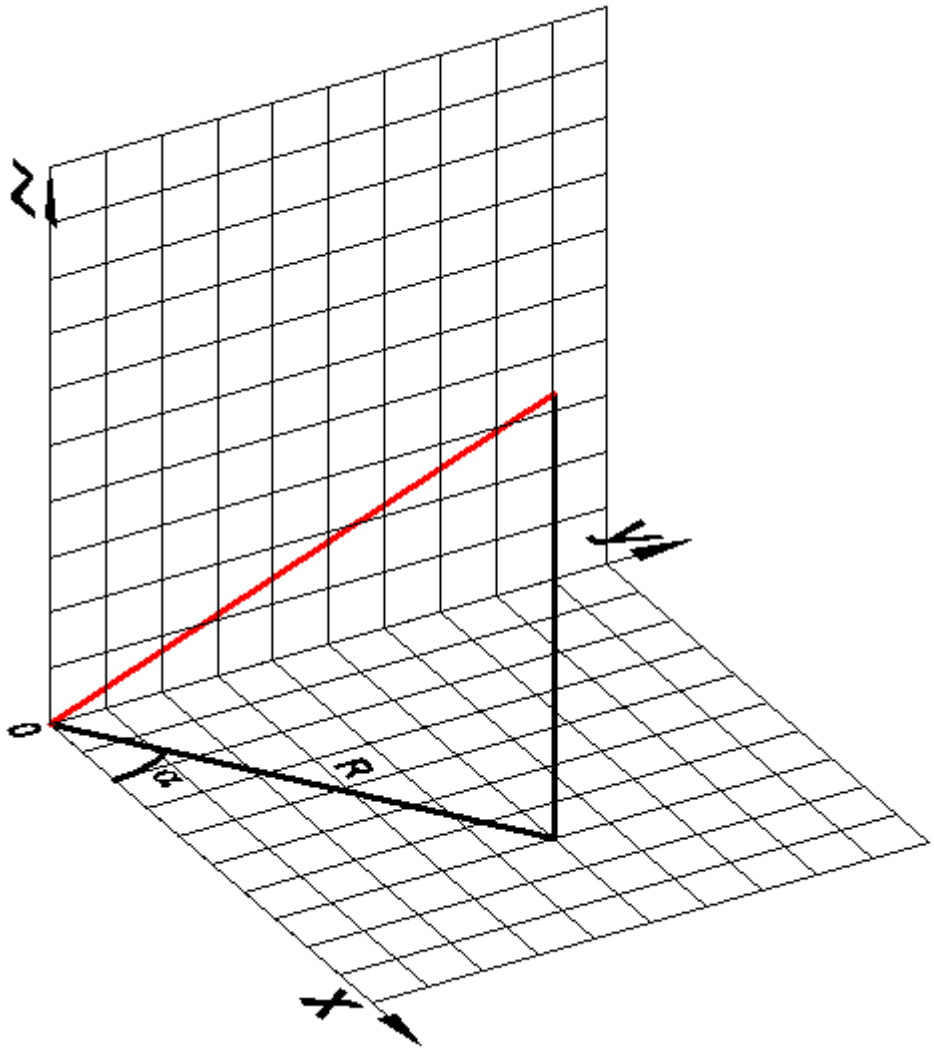
In a cylindrical coordinate system we use three perpendicular axis: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x- and y-axes define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined using the following format: R<alpha, z.

R = distance to the origin in the xy-plane

<alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)

z = the height above the xy-plane.

If the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.



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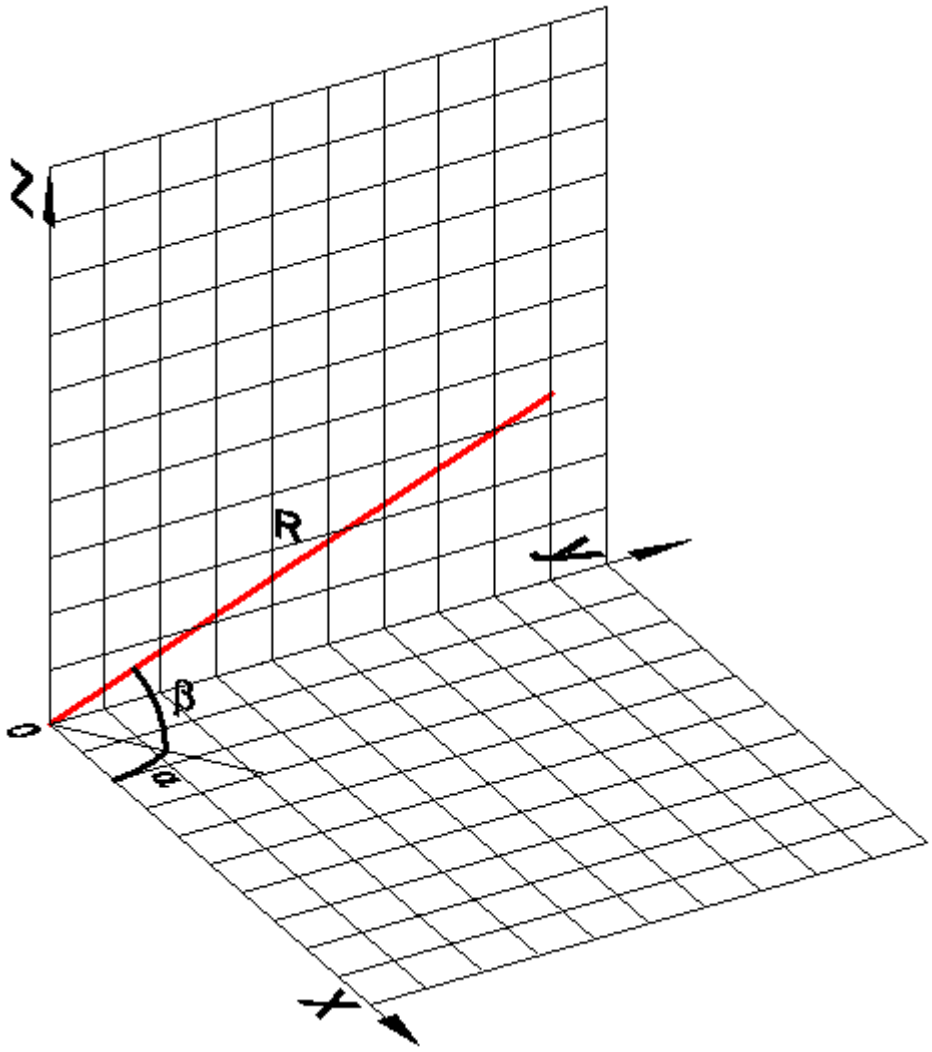
Working with spherical coordinates

In a cylindrical coordinate system we use three perpendicular axis: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x- and y-axes define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined using the following format: R<alpha<beta

R = distance from the origin

<alpha = angle in the xy-plane (positive angles are measured counter clockwise)

<beta = angle measured from the xy-plane (positive angles are measured above the xy-plane)



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Snap and Grid

Using Snap and Grid

Grid and snap help you to draw fast and accurately. A *grid* is a set of evenly spaced, visible dots that serve as a visual distance reference. The grid also indicates how far the drawing limits extend. The *snap* feature creates a set of evenly spaced, invisible magnetic points, which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. *Snap* constrains the points that you can pick with the mouse.

- Both *snap* and *grid* can be toggled on/off separately, giving you the opportunity to display the grid points, while snap is not active and vice versa.
- Both *snap* and *grid* can be set differently in each viewport.
- In each viewport you can rotate the grid using the Snap Angle setting.

Isometric Snap and Grid

You can use the Isometric snap and grid option to create two-dimensional isometric drawings. With the isometric option, you can draw a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings.

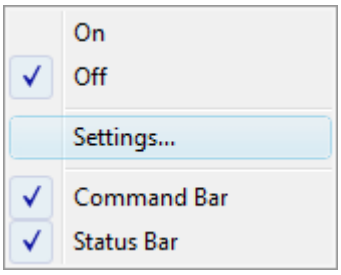
The isometric option always uses one of three preset planes, which are denoted as *Left*, *Right* and *Top*. You cannot alter the arrangement of these planes. If the Snap Angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

When you set the *Snap Style* setting to I*sometric Snap* and then set the *Snap Isometric Pair* setting to *Left*, *Top* or *Right*, the snap intervals, grid, and crosshairs align with the selected plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If the Orthogonal Mode is active, the movement of the crosshairs is constrained to the current isometric plane.

Display the Snap and Grid settings

Do one of the following to display the *Snap/Grid* settings in the *Settings* dialog:

- In the Status Bar, right click on the *Snap* field, then choose *Settings* in the context menu.

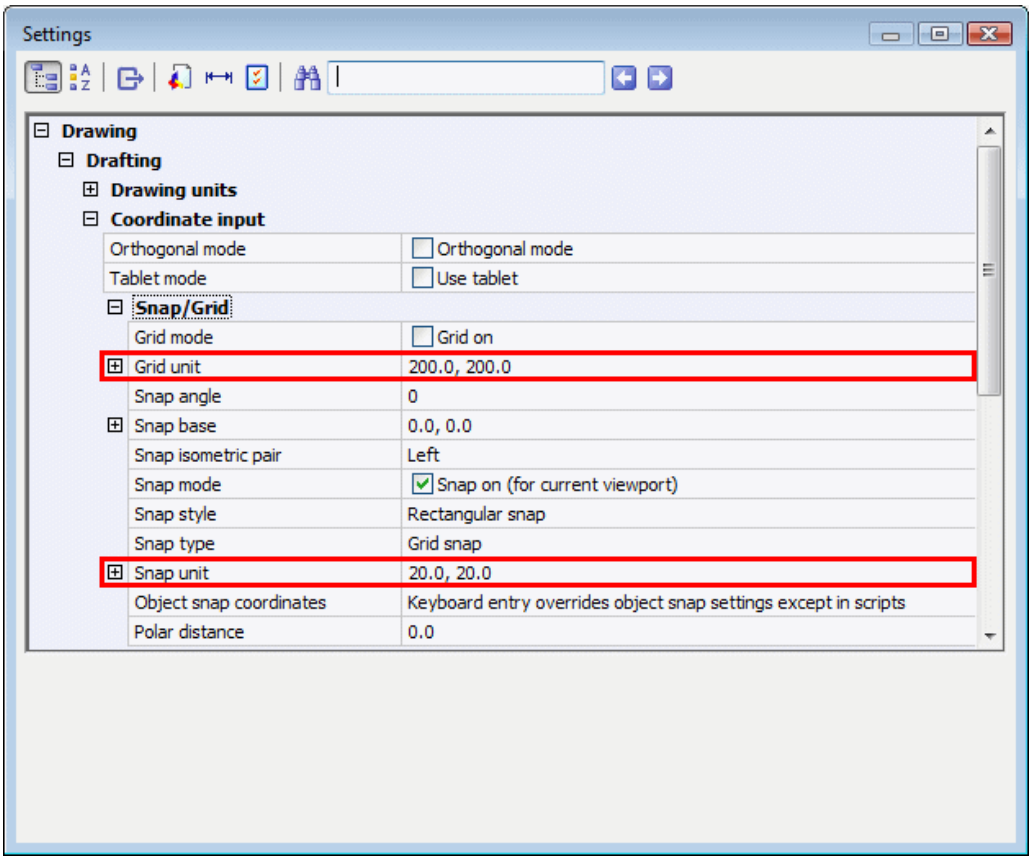


- Open the Systems Variables dialog, then click the Drawings button ( ) on the *Systems Variables* dialog.

  Under *Drafting*, expand *Coordinate Input* and *Snap/Grid*.

Defining snap and grid spacing

1. Display the *Snap/Grid* settings in the *Settings* dialog:



2. Set the *Reference Grid* in the current viewport:

- Expand the *Grid Unit* variable.
- Type a value in the X and Y fields

Grid unit	200.0, 200.0
X	200.0
Y	200.0

3. Set the *Snap Spacing*:

- Expand the *Snap Unit* variable.
- Type a value in the X and Y fields

Snap unit	20.0, 20.0
X	20.0
Y	20.0

4. Close the *System Variables* dialog window.
5. If the *Grid* display is not turned on yet, do one of the following:

- Double click the *GRID* field (GRID) in the *Status Bar*.
- Click the Grid tool ( ) on the *Settings* toolbar.

6. If *Snap* is not turned on yet, do one of the following:

- Double click the *SNAP* field (SNAP) in the *Status Bar*.
- Click the Snap tool ( ) on the *Settings* toolbar.

NOTES

- Although not necessary, it is recommended to set the *Reference Grid* as a multiple of the *Snap Spacing.*
- The *Reference Grid* only displays within the *Drawing Limits*. If necessary adjust the *Drawing Limits*.

Setting the drawing Limits

1. Do one of the following:

- Click the Drawing Limits tool button () on the *Settings* toolbar.
- Choose Drawing Limits in the *Settings* menu.
- Type *limits* in the command bar, then press Enter.

The command bar reads: Limits are off: ON/<Lower left corner> <x,y>:

2. Do one of the following:

- Press Enter to accept the current lower left corner.
- Specify the lower left corner of the drawing limits.

The command bar reads: Upper right corner <x,y>:

3. Do one of the following:

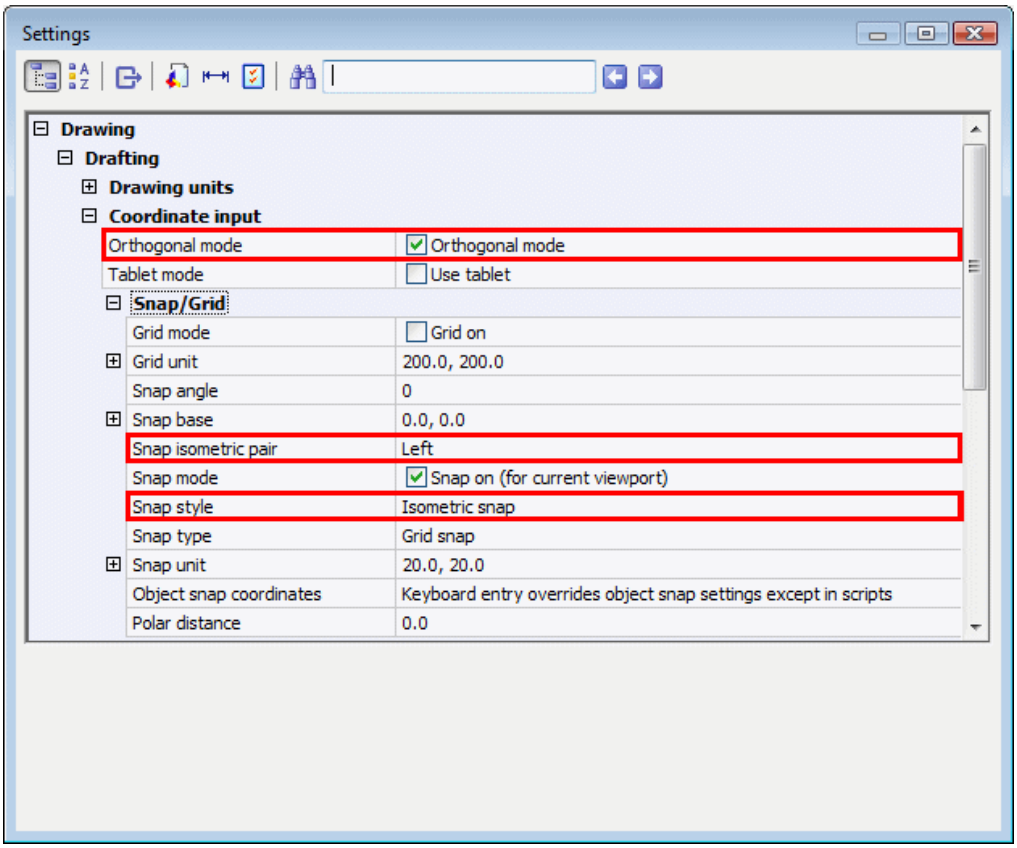
- Press Enter to accept the current upper right corner.
- Specify the upper right corner of the drawing limits.

Command Options

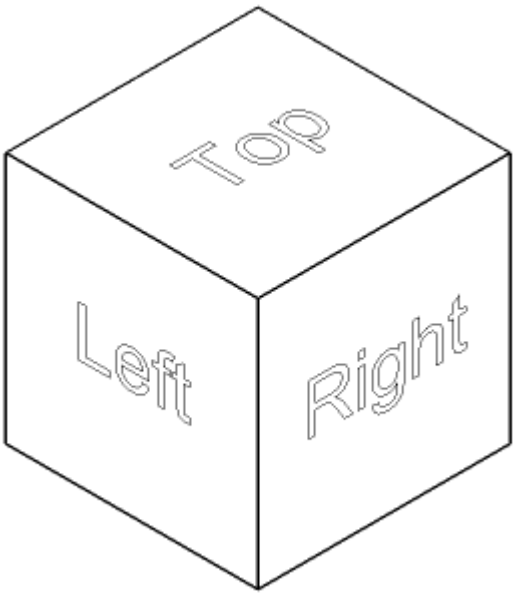
Keyboard	Prompt Box	Description
ON	Turn limits on	Turns the drawing limits on. When the drawing limits are on it is impossible to draw outside of the limits.
OFF	Turn limits off	Turns the drawing limits off.
Esc	Cancel	Aborts the Limits command.

Using Isometric Snap

1. Display the *Snap/Grid* settings in the *Settings* dialog:



2. Turn on the *Orthogonal Mode*.
3. Set the *Snap Style* setting to *Isometric Snap*.
4. In *Snap isometric pair*, set the appropriate drawing plane: *Top*, *Left* or *Right*.



Setting the Snap Angle

1. Display the *Snap/Grid* settings in the *Settings* dialog:
2. Select the *Snap Angle* setting.
3. Type a new value in the *Snap Angle* setting field.
4. Close the *Settings* dialog.

NOTE You can also set the *Snap Angle* by typing *snapang* in the command bar.

When in a command, type *'snapang* (with an apostrophe in front), to set the *Snap Angle* variable transparently (= without interrupting the running command).

Entity Snaps

Entity Snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

When you specify one or more entity snaps, the entity *Snap Aperture Box* is added to the crosshairs. In addition, an icon appears adjacent to the crosshair indicating the active entity snap (*Snap cursor decoration*). When you move the cross hairs, the program snaps to the snap point closest to the center of the *Snap Aperture Box*. The *Snap Marker* indicates the current snap point. Press the TAB key to cycle through all possible entity snaps.

To define the Entity Snap settings

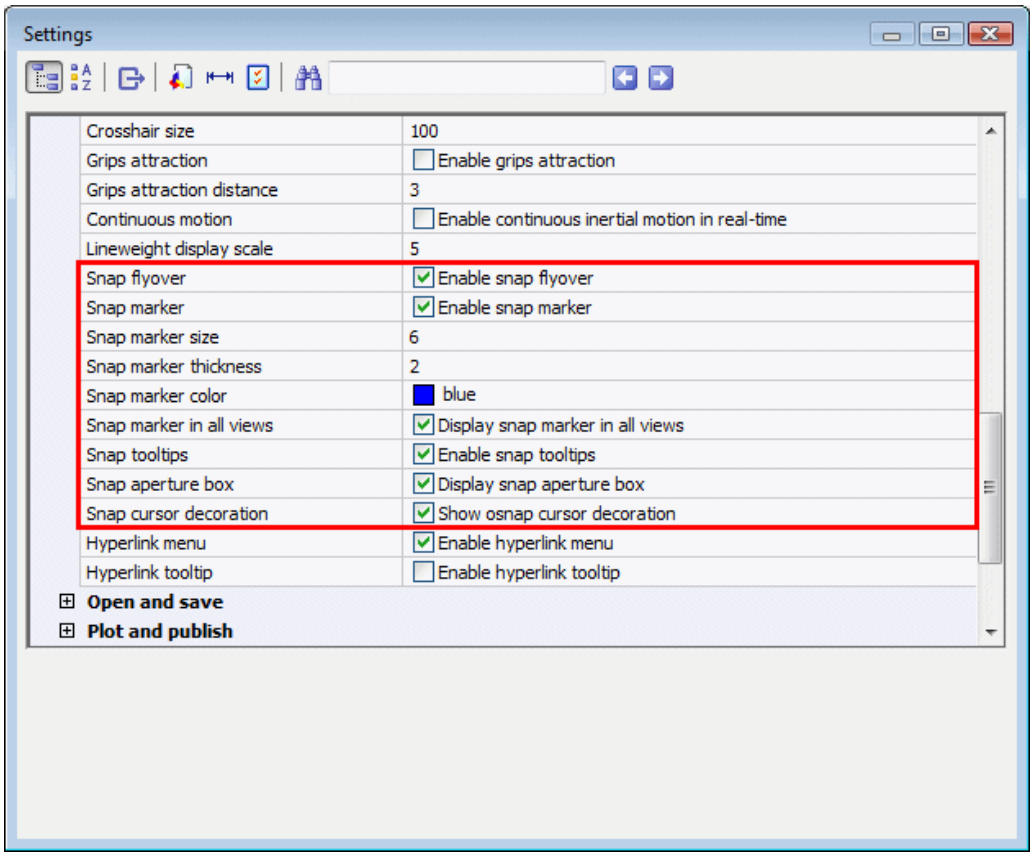
1. Do one of the following:

- Click the Settings tool button () on the Settings toolbar.
- Choose *Settings* in the *Settings* menu.
- Type *settings* in the command bar, then press Enter.

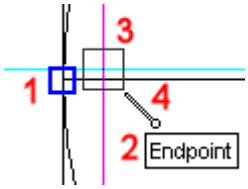
The *Settings* dialog opens.

2. In the *Settings* dialog, expand the *Program Options* settings class.

3. Under *Program Options* expand the *Display* settings group, then scroll down to the *Entity Snap* settings.



4. Define the *Entity Snap* settings.



Setting	Description
Snap flyover	Enables Entity Snap.
Snap marker (1)	Enables the display of the snap marker.
Snap marker size	Sets the size of the snap marker. (default size is 6)
Snap marker thickness	Sets the thickness of the snap marker. (default thickness is 2)
Snap marker color	Sets the color of the snap marker.
Snap marker in all views	If multiple viewports are open, enables the display of the snap marker in all viewports.
Snap tooltips (2)	Enables the display of the Entity Snap tooltips.
Snap aperture box (3)	Sets the size of the Entity Snap aperture box. (default size is 10)
Snap cursor decoration (4)	Enables the display of the current Entity Snap icon adjacent to the cross hairs.

NOTE You can toggle the *Entity Snaps* on/off by double clicking the *ESNAP* field in the Status Bar.

To set the Entity Snaps

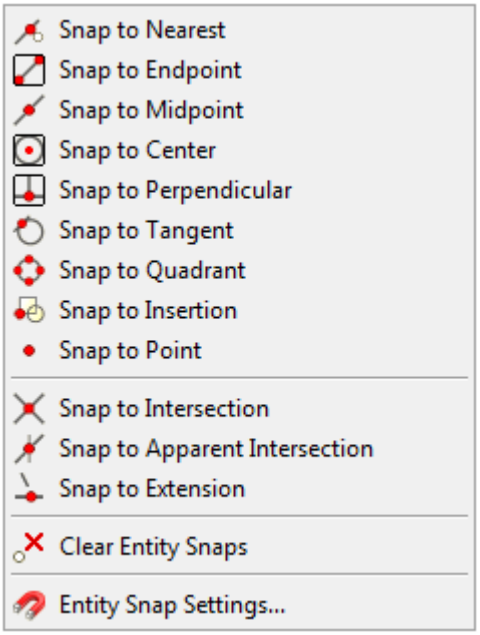
Do one of the following:

- Click the buttons on the *Entity Snaps* toolbar.

  The buttons of the currently active Entity Snap modes are pressed.

- Press and hold the Shift key, then right click and select the entity snap mode in the context menu.

  The icons of the currently active Entity Snap modes are outlined.



When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Entity Snap Modes

Name	Icon	Marker	Description
Nearest			Snap to the nearest point on an entity.
Endpoint			Snap to the nearest endpoint of an entity or polyline segment.

Midpoint			Snap to the midpoint of an entity or polyline segment.
Center			Sncap to the center point of an arc, circle, polygon, ellipse or elliptical arc. Snap to the center of gravity of a closed polyline.
Perpendicular			Snap to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
Tangent			Snap to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.
Quadrant			Snap to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
Insertion			Snap to the insertion point of an attribute, block or text entity.
Point			Snap to a point entity.
Intersection			Snap to the intersection of any combination of entities.
Apparent Intersection			Snap to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space.
Extension			Snaps to the extension of an entity or to the intersection of the extension of two entities.
Clear			Turn off all entity snap modes.

Working with multiple Entity Snap modes

1. Move the cursor to the entity you want to snap.
   One of the active Entity Snap modes markers display.
2. Press the TAB key.
   The entity the snap point is on highlights.
3. Do one of the following:

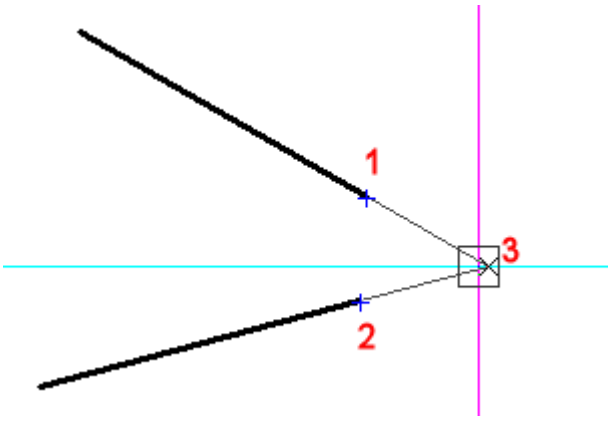
- Click to accept the snap point.
- Press the TAB key.
  The next possible Entity Snap mode marker displays.
  The entity the snap point is on highlights.

4. (option) Repeat step 3 until the right snap point is found.

NOTE If you keep pressing the TAB key you can cycle through all possible snap points.

To snap to the extension of two entities

1. If not already on, turn on the *Extension* Entity Snap mode ( ).
2. Launch a drawing tool, e.g. Draw Line.
3. Move the cursor over the endpoint of the first entity (1).
   A small cross (+) indicates the entity is marked for extension.
4. Move the cursor over the endpoint of the second entity (2).
   A small cross (+) indicates the entity is marked for extension.
5. Move the cursor near the intersection of the extensions of the two entities.
   An X indicates the intersection (3).



6. Click to accept the snap point.

NOTE You can snap to the extension of lines, polylines, arcs and elliptical arcs

Using Orthogonal Mode

Orthogonal Mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the current coordinate system. For example, with the default 0-degree orientation (angle 0 at the "three o'clock" or "east" position), when the *Orthogonal Mode* setting is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor: type the length of the line in the command bar, then press Enter.

NOTES

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the *Orthogonal Mode* axes using the Snap Angle setting.
- Press and hold the Shift key to reverse the *Orthogonal Mode* setting.
- *Orthogonal Mode* is overruled by entity snaps.

To toggle Orthogonal Mode

Do one of the following:

- Double click the *ORTHO* field in the Status Bar.
- Press the *F8* function key.
- Type *ortho* in the command bar, then choose the appropriate setting.
- Set the *Orthogonal Mode* in the Settings dialog.

NOTE Turning on *Ortho Mode*, automatically disables Polar Tracking.

Polar Tracking

Polar Tracking

Polar tracking assists you to draw at exact angles:

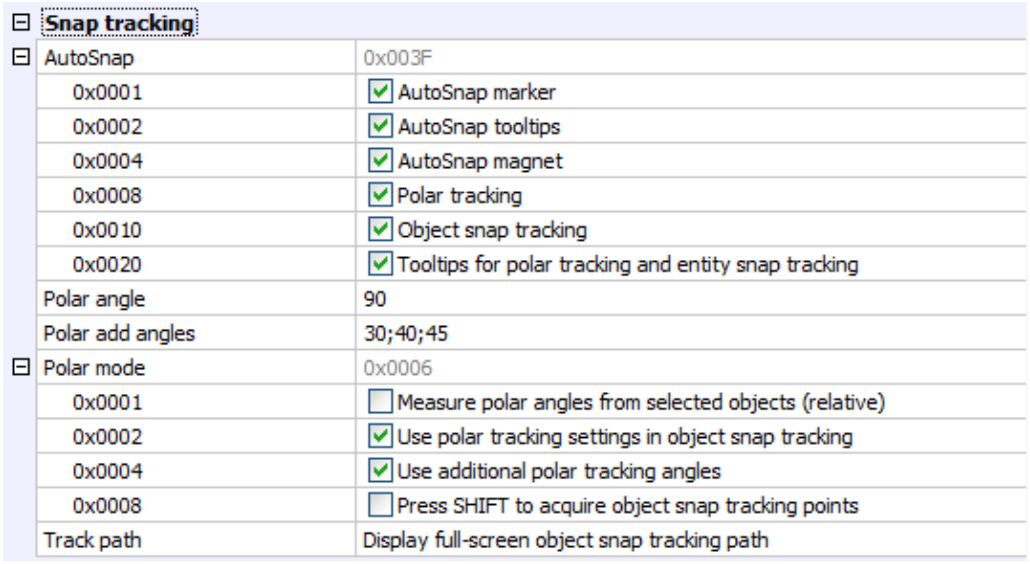
- at fixed intervals, starting from 3 O'clock (or East)
- at specific additional angles

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as the *Snap Aperture* box crosses a polar tracking line.

To define the Polar Tracking settings

1. Do one of the following:

- Right click on the *POLAR* field in the Status Bar, then right click and choose *Settings* in the context menu.
- Open the Settings dialog, then expand the *Coordinate input* sub-category under *Drafting* in the *Drawing* settings class.
  Expand the *Snap Tracking* settings group.

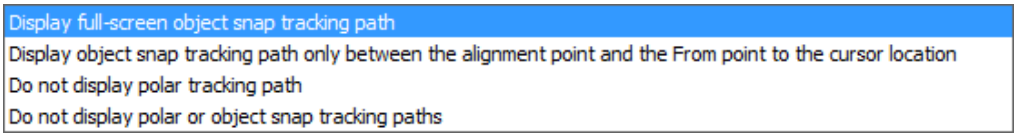


Snap tracking	
AutoSnap	0x003F
0x0001	☑ AutoSnap marker
0x0002	☑ AutoSnap tooltips
0x0004	☑ AutoSnap magnet
0x0008	☑ Polar tracking
0x0010	☑ Object snap tracking
0x0020	☑ Tooltips for polar tracking and entity snap tracking
Polar angle	90
Polar add angles	30;40;45
Polar mode	0x0006
0x0001	☐ Measure polar angles from selected objects (relative)
0x0002	☑ Use polar tracking settings in object snap tracking
0x0004	☑ Use additional polar tracking angles
0x0008	☐ Press SHIFT to acquire object snap tracking points
Track path	Display full-screen object snap tracking path

2. Adjust the settings:

- Autosnap > 0x0001: Autosnap marker (not used, required for AutoCad® compatibility only)
- Autosnap > 0x0002: Autosnap tooltips (not used, required for AutoCad® compatibility only)
- Autosnap > 0x0004: Autosnap magnet (not used, required for AutoCad® compatibility only)
- Autosnap > 0x0008: If checked, *Polar Tracking* is on.
- Autosnap > 0x0010: If checked, *Object Snap Tracking* is on.

- Autosnap > 0x0010: If checked, tooltips display for polar tracking and entity snap tracking.
- Polar angle: Sets the polar angle increment.
- Polar add angles: Sets the additional polar angles. Separate angles by semicolons.
- Polar mode > 0x0001: If checked, polar angles are measured relatively from selected entities.
- Polar mode > 0x0002: If checked, the polar angles (interval and additional angles) are also used in entity snap tracking.
- Polar mode > 0x0004: If checked, the additional angles for polar tracking are used.
- Polar mode > 0x0008: If checked, press and hold the Shift key to acquire entity tracking points.
- Track path: Controls the display of the polar and entity snap tracking paths.



Using Polar Tracking to draw a line

1. (option) Check the Polar Tracking settings.
2. (option) If not already on, double click the *POLAR* field in the Status Bar.
3. Launch the Draw Line tool.
4. Specify the start point of the line.
   Polar tracking lines display at the specified intervals and additional angles.
   A cross marker (X) indicates the current point.
5. Do one of the following to define the endpoint of the line.
   When the appropriate tracking line displays:

- Click when the cursor is at the desired position.
- Use Direct Distance Entry: type the length of the line in the command bar and press Enter.

NOTE Turning on *Polar Tracking* automatically disables Orthogonal Mode.

Entity Snap Tracking

Entity Snap Tracking assists you to draw entities in relationship to other entities.

When *Entity Snap Tracking* is turned on, Bricscad displays temporary alignments based on entity snap points. *Entity Snap Tracking* works in combination with Entity Snap tools. At least one *Entity Snap* mode must be on if you want to use *Entity Snap Tracking.*

Entity Snap Tracking alignments are parallel to the X- and Y-axis of the current UCS by default. If the Polar mode > 0x0002 setting is checked, the polar angles used in Polar Tracking (interval and additional angles) are also used in *Entity Snap Tracking*.

Alignment points must be acquired by moving the cursor over entity snap points. Acquired points display a small plus sign (+). To remove an acquired point, move the cursor back over the plus sign (+). If the Polar mode > 0x0008 is checked, you must press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their drawing paths. You can then define points either on the alignments or at the intersection of two alignments.

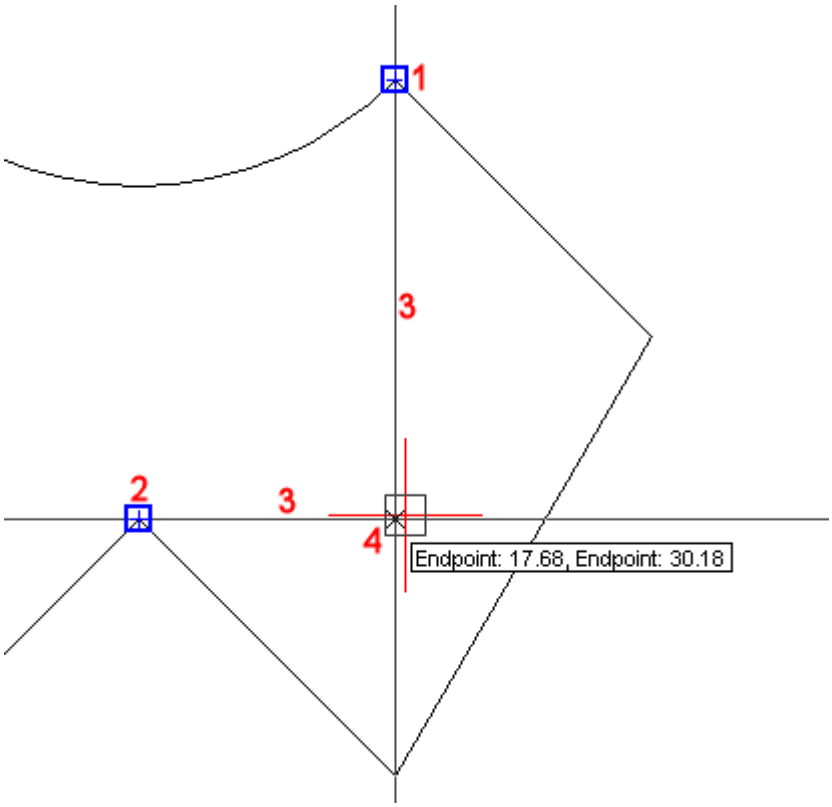
Using Entity Snap Tracking

1. Launch a drawing tool, e.g. Draw Line.
2. Hover the cursor over the first tracking point (1).

   A small plus sign (+) displays to indicate the tracking point is active.
3. Hover the cursor over the second tracking point (2)

   A small plus sign (+) displays to indicate the tracking point is active.
4. Move the cursor close to position 4.

- Tracking lines display from both tracking points (3).
- A cross marker (X) indicates the intersection of the tracking lines.
- Snap markers display at the tracking points.
- An *Entity Snap Tracking* tooltip shows the distances from the tracking points.



5. Click to accept the *Entity Snap Tracking* point.

User Coordinate Systems

User Coordinate Systems

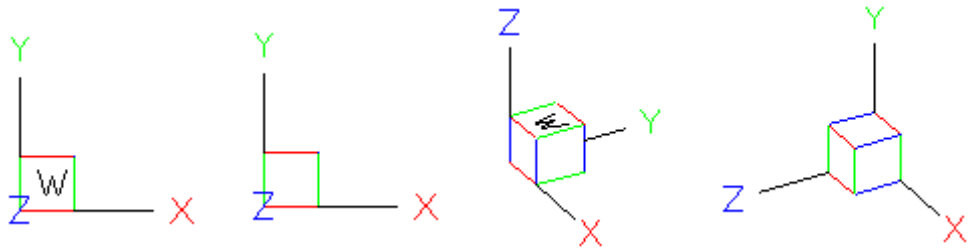
When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction. You can create as many UCS as you want, saving or redefining them to help you construct three-dimensional entities. By defining a UCS within the WCS, you can simplify the creation of most three-dimensional entities into combinations of two-dimensional entities.

When you begin a new drawing, you are automatically in the WCS, indicated by the letter W in the icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect your new viewpoint.

NOTE You cannot delete or modify the WCS.

To help you keep your bearings in the current coordinate system, the Bricscad displays a coordinate system icon. The visible portions of the axes are the positive directions.



WCS Top View | UCS Top View | WCS Isometric View | UCS Isometric View

Three colors represent the three axes, making it easier for you to recognize the orientation in three-dimensional space:

- x-axis: red
- y-axis: green
- z-axis: blue

The Plan View command restores the Plan view or Top view of the current UCS or WCS.

The UCSICON variable controls the display and location of the UCS icon:

UCSICON	0x0003
0x0001	Show icon
0x0002	at origin

Show icon:

Controls whether the icon shows or not.

at origin:

Controls the location of the UCS icon: if on, the icon indicates the origin point of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport borders, the UCS icon moves to the bottom right corner of the viewport.
When the *at origin* option is not checked, the icon always displays in the bottom right corner of the viewport.

To define a User Coordinate System

1. Do one of the following:

- Click the *Coordinate Systems...* tool button ( ) on the Settings toolbar.
- Choose *Coordinate Systems* in the *Settings* menu.
- Type *expucs* in the command bar, then press Enter.

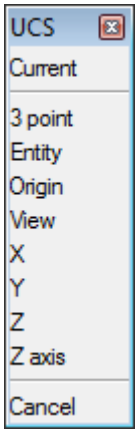
The *Bricscad Explorer - Coordinate Systems* dialog opens.

2. Click the *New* tool button ( ) on the *Bricscad Explorer* dialog.

The *Bricscad Explorer* dialog closes.

The command bar reads: Current/Entity/Origin/View/X/Y/Z/ZAxis<3point>:

A prompt box displays:



3. Press Enter to define the UCS using points.

The command bar reads: New origin <current origin>:

4. Specify the origin point of the UCS.

The command bar reads: Point of positive X axis <current point>:

5. Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

6. Specify a point to define the positive Y-axis.

The UCS is defined.

The *Bricscad Explorer* dialog reopens.

7. Click in the *UCS Name* field of the newly defined UCS to replace the <NewUCS> default name.

8. Close the *Bricscad Explorer* dialog.

Command Options

Keyboard	Prompt Box	Description
C	Current	Save the current coordinate system.
Enter	3 point	Define the coordinate system using points.
E	Entity	Align the coordinate system with an entity.
O	Origin	Define the coordinate system parallel to the current coordinate system.
V	View	Align the coordinate system with the current view orientation (1).
X	X	Rotate the current coordinate system about its X-axis.
Y	Y	Rotate the current coordinate system about its Y-axis.
Z	Z	Rotate the current coordinate system about its Z-axis.
ZA	Z axis	Define the Z-axis of the new coordinate system (2).
Esc	Cancel	Abort the command.

(1) X-axis is parallel to the bottom edge of the screen and Z-axis perpendicular to the view orientation, positive Z-axis pointing to the viewer. The origin is copied from the previous coordinate system.

(2) The first point defines the origin of the UCS. The econd 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.

To restore the WCS

1. Type UCS in the command bar, then press Enter.
   The command bar reads:
   ?/3point/Delete/Entity/Origin/Previous/Restore/Save/View/X/Y/Z/Zaxis/<World>:
   A prompt box displays.
2. Do one of the following:

- Press Enter to accept the default option.
- Choose *World* in the prompt box.
- Type *W* in the command bar and press Enter.

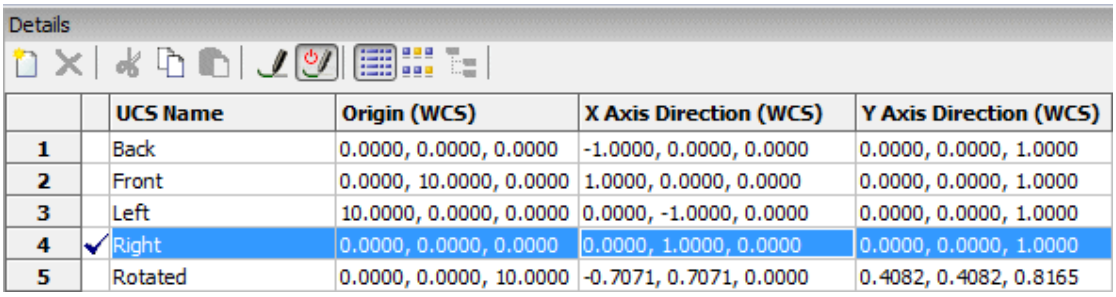
To restore a UCS

1. Do one of the following:

- Click the *Coordinate Systems...* tool button () on the Settings toolbar.
- Choose *Coordinate Systems* in the *Settings* menu.
- Type *expucs* in the command bar, then press Enter.

  The *Bricscad Explorer - Coordinate Systems* dialog opens.

2. Click the blank tile in front of the UCS name to make it current.



		UCS Name	Origin (WCS)	X Axis Direction (WCS)	Y Axis Direction (WCS)
1		Back	0.0000, 0.0000, 0.0000	-1.0000, 0.0000, 0.0000	0.0000, 0.0000, 1.0000
2		Front	0.0000, 10.0000, 0.0000	1.0000, 0.0000, 0.0000	0.0000, 0.0000, 1.0000
3		Left	10.0000, 0.0000, 0.0000	0.0000, -1.0000, 0.0000	0.0000, 0.0000, 1.0000
4	✓	Right	0.0000, 0.0000, 0.0000	0.0000, 1.0000, 0.0000	0.0000, 0.0000, 1.0000
5		Rotated	0.0000, 0.0000, 10.0000	-0.7071, 0.7071, 0.0000	0.4082, 0.4082, 0.8165

3. Close the *Bricscad Explorer* dialog.

NOTE If no UCS is marked, the WCS is the current coordinate system.

Direct Distance Entry

Direct distance entry (DDE) means that you can type the length of lines, polyline segments, circle radius and diameter, movement vector in commands such as Move, Copy and Stretch, etc. in the command bar.

DDE can be used whenever Orthogonal Mode or Polar Tracking is active.

To help you keep track of the position of the cursor it is recommended to set the readout of the coordinate field in the Status Bar to "Coordinates in polar form for point, distance and angle selection".

To define the Coordinates setting

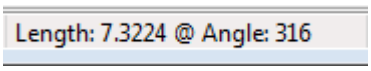
Do one of the following:

- Type *coords* in the command bar, then press Enter.

  Type 2 and press Enter.

- Open the Settings dialog and expand the *Display/Viewing* settings sub-category under *Drafting* in the *Drawing* settings class.

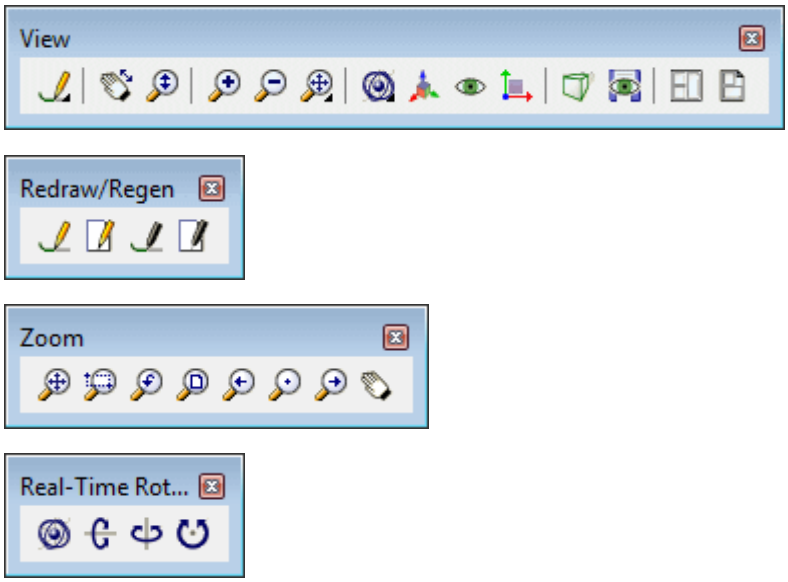
  Set the *Coordinates* setting to *Coordinates in polar form for point, distance and angle selection*.



Viewing your drawing

Overview

Bricscad provides various tools to control the display and view orientation of your drawing. These tools can be found either in the *View* menu or the *View* toolbar or both.



View toolbar and flyouts

Icon	Tool name	Keyboard	Description
	Redraw	redraw	Refreshes the screen display in the current viewport.
	Redraw All	redrawall	Refreshes the screen display in all open viewports.
	Regenerate	regen	Recalculates the screen display in the current viewport.
	Regenerate All	regenall	Recalculates the screen display in all open viewports.
	Real Time Pan	rtpan	Pans the view dynamically.
	Pan	pan	Pans the view.
	Real Time Zoom	rtzoom	Zooms in/out dynamically.
	Zoom In	zoom I(*)	Zooms in on the center of the window by a factor of 2.
	Zoom Out	zoom O(*)	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents	zoom E(*)	Displays all the entities in the drawing (referred to as the *drawing extents*).

	Zoom Window	zoom W(*)	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous	zoom P(*)	Restores the displayed view prior to the current one.
	Zoom All	zoom A(*)	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	zoom L(*)	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	zoom C(*)	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	zoom R(*)	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Real-Time Sphere	rtrot	Rotates the view dynamically in 3D.
	Real-Time X	rtrotx	Rotates the view dynamically about the screen x-axis.
	Real-Time Y	rtroty	Rotates the view dynamically about the screen y-axis.
	Real-Time Z	rtrotz	Rotates the view dynamically about the screen z-axis.
	Dynamic View Control	viewctl	Opens the *View Control - Relative to WCS* dialog.
	Set View Point	setvpoint	Opens the *Preset Viewpoints* dialog.
	Plan View	plan	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	dview	Defines parallel and visual perspective views.
	Save/Restore view	view	Saves and restores named views.
	Viewports	viewports	Manages viewports in model space.
	Paper Space Views	mview	Creates viewports in paper space.

(*) <command> <option>: type *zoom* followed by Enter, then type the command option letter, followed by Enter.

View manipulation using the mouse

View manipulation using the mouse

In Bricscad you can navigate through your drawing using the mouse.

Mouse / Key	Action	Result
Mouse Wheel	scroll	Zoom in / out (*)
Middle Button (**)	press and hold while moving the mouse	Real-time pan
Middle Button (**)	double click	Zoom extents
Ctrl + Shift + Left Button	press and hold while moving the mouse	Zoom in / out
Ctrl + Shift + Right Button	press and hold while moving the mouse	Real-time pan
Ctrl + Left Button	press and hold while moving the mouse	Real-time sphere
Ctrl + Right Button	press and hold while moving the mouse	Rotate about the screen Z-axis

(*) The incremental change in zoom with each mouse-wheel action is controlled through the *Zoom Factor* variable (ZOOMFACTOR).

(**) On condition the *Middle Button Pan* (MBUTTONPAN) variable is set to *On*.

Setting the Zoom Factor variable

1. Type *zoomfactor* in the command bar, then press Enter.
   The command bar reads: New current value for ZOOMFACTOR (3 to 100) <current value>:
2. Type a value between 3 and 100 in the command bar, then press Enter.

or

1. In the System Variables dialog, open the *Drawing* () category, then expand the *Display/Viewing* sub-category.
   In the *Viewing* group, select the ZOOMFACTOR variable.
2. Type a value between 3 and 100 in the *zoomfactor* field.

NOTE Increase the zoom factor to speed up scroll wheel zooming in large drawings.

Setting the Middle Button Pan variable

1. Type in *mbuttonpan* the command bar, then press Enter.
   The command bar reads: New current value for MBUTTONPAN (Off or On) <current value>:
2. Type *On* or *Off* in the command bar.

or

1. In the System Variables dialog, open the *Program Options* () category, then expand the *User Preferences* sub-category and select the MBUTTONPAN variable.
2. Choose either *Support panning* or *Support action defined in menu file*.

Redrawing and Regenerating a Drawing

Redrawing and Regenerating a Drawing

As a matter of fact the screen display of a drawing is a simplified version of the drawing database. From time to time it is necessary to synchronise the screen display and the drawing database. Most of the time Bricscad regenerates the display automatically, but in a few cases a forced regeneration of the drawing might be necessary. This is done by the *Regen* command.

Don't confuse the *Regen* command with the *Redraw* command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

Regenerate the current viewport

Do one of the following:

- Click the Regen tool button () on the *View* toolbar.
- Choose Regen in the *View* menu.
- Type *re* or *regen* in the command bar, then press Enter.

The current viewport is regenerated.

Regenerate all viewports

Do one of the following:

- Click the Regenall tool button () on the *View* toolbar.
- Type *rea* or *regenall* in the command bar, then press Enter.

All viewports are regenerated.

Redraw the current viewport

Do one of the following:

- Click the Redraw tool button () on the *View* toolbar.
- Choose Redraw in the *View* menu.
- Type *r* or *redraw* in the command bar, then press Enter.

The current viewport is refreshed.

Redraw all viewports

Do one of the following:

- Click the Redraw All tool button () on the *View* toolbar.
- Type *ra* or *redrawall* in the command bar, then press Enter.

All viewports are refreshed.

Panning

Panning

The *Pan* command moves the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed. The cursor changes to a hand ( ) when a *Pan* tool is

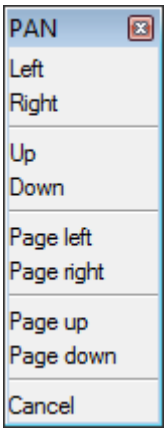
Using the Pan command

1. To launch the Pan command do one of the following:

- Click the Pan tool button ( ) on the *View* toolbar (*Zoom Extents* flyout).
- Choose Pan in the *View* menu.
- Type *p* or *pan* in the command bar, then press Enter.

The command bar reads: Left/Right/Up/Down/PGLeft/PGRight/PGUp/PGDown/<Pan base point>:

A prompt box opens:



2. Specify the *Pan base point*.

The command bar reads: Pan displacement point:

3. Specify the *Pan displacement point*.

The display shifts over the specified distance and in the specified direction.

Command Options

Keyboard	Prompt Box	Description
L	Left	Shifts the view one step to the left.
R	Right	Shifts the view one step to the right.
U	Up	Shifts the view one step up.
D	Down	Shifts the view one step down.
PGL	Page left	Shifts the view one screen to the left.

PGR	Page right	Shifts the view one screen to the right.
PGU	Page up	Shifts the view one screen up.
PGD	Page down	Shifts the view one screen down.

Using real time panning

1. To launch the Real-time Pan command do one of the following:

- Click the Real-Time Pan tool button () on the *View* toolbar
- Choose *Real-Time Motion* > *Real-Time Pan* in the View menu.
- Type *rtpan* in the command bar, then press Enter.

The command bar reads: >>ENTER, Right click or Esc to complete...

2. Press and hold the left mouse button to pan the view.
3. To abort the *Real-Time Pan* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

Zooming

Zooming

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass ( ) when a *Zoom* tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

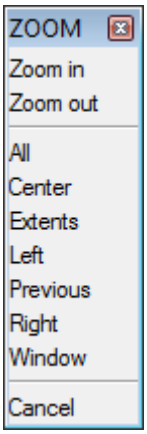
Using the Zoom command

1. Do one of the following:

- Type *zoom* in the command bar.
- Type *Z* in the command bar.

The command bar reads: In/Out/All/Center/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>:

A prompt box opens:



2. Do one of the following:

- Type the zoom factor, followed by *X*. E.g. type *2x* to magnify the display 2 times; typing *0.5x* changes the display to half its original size.
- In paper space viewports, type the zoom factor followed by *XP* to define the zoom factor relative to the viewport, thus defining the scale of the viewport content.
- Pick two corners of a box on the existing view in order to enlarge that area to fill the display.

Command Options

Keyboard	Prompt Box	Description
I	Zoom in	Zooms in on the center of the window by a factor of 2 ( )
O	Zoom out	Zooms out from the center of the window by a factor of 0.5 ( )

A	All	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two) ()
C	Center	Pick the center of the next view, then specify the magnification factor or the view height ()
E	Extents	Displays all the entities in the drawing (referred to as the *drawing extents*) ()
L	Left	Pick the lower left corner of the next view, then specify the magnification factor or the view height ()
P	Previous	Restores the displayed view prior to the current one ()
R	Right	Pick the upper right corner of the next view, then specify the magnification factor or the view height ()
W	Window	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display ()

Real-time zooming

1. Do one of the following:

- Click the Real-Time Zoom tool button () in the View toolbar.
- Type *rtzoom* in the command bar, then press Enter.

The command bar reads: >>ENTER, Right click or Esc to complete...

2. Press and hold the left mouse button.

Move the mouse forward to zoom in.

Move the mouse backward to zoom out.

3. To abort the *Real-Time Zoom* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

View Rotation

View Rotation

In order to view your drawing from any angle, you can rotate a view. The *Real-Time Motion* tools of Bricscad allow you to rotate a view in real-time. You can rotate the view about the X, Y or Z screen axis or in any direction (real-time sphere). If the *Continuous Motion* variable is set, the view rotation continues until you conclude the *Real-Time Motion* command.

NOTES

- Real-time Sphere, Real-time X and Real-time Y should not be used when drawing in 2D.
- Use the Plan View tool to restore top view.

Rotating a view freely

1. Do on of the following:

- Click the Real-Time Sphere tool button () on the *View* toolbar.
- Choose *Real-Time Motion* > *Real-Time Sphere* in the *View* menu.
- Type *rtrot* in the command bar, then press Enter.

The command bar reads: >> ENTER, Right click or Esc to complete ...

2. Press and hold the left mouse button.

Move the mouse to rotate the view in any direction.

3. To abort the *Real-Time Sphere* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

Go to top

Rotating a view about the view X-axis

1. Do on of the following:

- Click the Real-Time X tool button () on the *View* toolbar.
- Choose *Real-Time Motion* > *Real-Time X* in the *View* menu.
- Type *rtrotx* in the command bar, then press Enter.

The command bar reads: >> ENTER, Right click or Esc to complete ...

2. Press and hold the left mouse button.

Move the mouse to rotate the view.

3. To abort the *Real-Time X* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

Go to top

Rotating a view about the view Y-axis

1. Do on of the following:

- Click the Real-Time Y tool button () on the *View* toolbar.
- Choose *Real-Time Motion* > *Real-Time Y* in the *View* menu.
- Type *rtroty* in the command bar, then press Enter.

The command bar reads: >> ENTER, Right click or Esc to complete ...

2. Press and hold the left mouse button.

Move the mouse to rotate the view.

3. To abort the *Real-Time Y* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

Go to top

Rotating a view about the view Z-axis

1. Do on of the following:

- Click the Real-Time Z tool button () on the *View* toolbar.
- Choose *Real-Time Motion* > *Real-Time Z* in the *View* menu.
- Type *rtrotz* in the command bar, then press Enter.

The command bar reads: >> ENTER, Right click or Esc to complete ...

2. Press and hold the left mouse button.

Move the mouse to rotate the view.

3. To abort the *Real-Time Z* command, do one of the following:

- Right click.
- On the keyboard, press Enter, space bar or Esc.

Go to top

Using Dynamic View Control

1. Do one of the following:
2. (under construction)

Using Preset Viewpoints

1. Do one of the following:
2. (under construction)

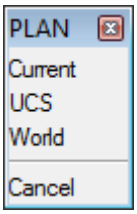
Isometric Views toolbar.

Restoring Plan View

1. Do one of the following:

- Click the Plan View tool button () on the *View* toolbar.
- Choose *Plan View* in the *View* menu.
- Type *plan* in the command bar, then press Enter.

  The command bar reads: Plan view of: UCS/World/<current UCS>:
  A prompt box opens:



2. Press Enter to restore the plan view with respect to the current coordinate system.

Command Options

Keyboard	Prompt Box	Description
Enter	Current	(default command option) Restore the plan view with respect to the current coordinate system.
U	UCS	Show the plan view of a saved User Coordinate System. You are prompted to type the name of a UCS.
W	World	Show the plan view with respect to the World Coordinate System (WCS).

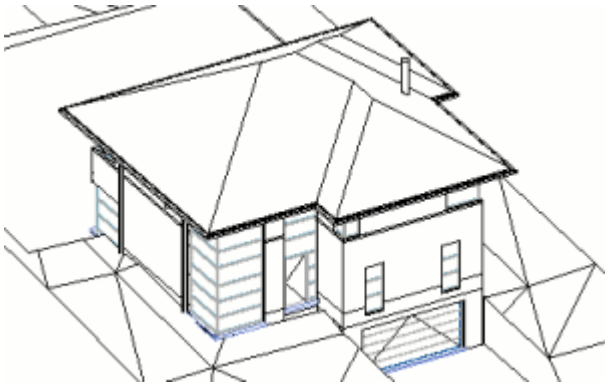
NOTES

- If the WCS is the current coordinate system, the *Current* and *World* options have the same result.
- If the UCSFOLLOW variable is set to *ON*, the plan view is generated whenever the UCS changes.

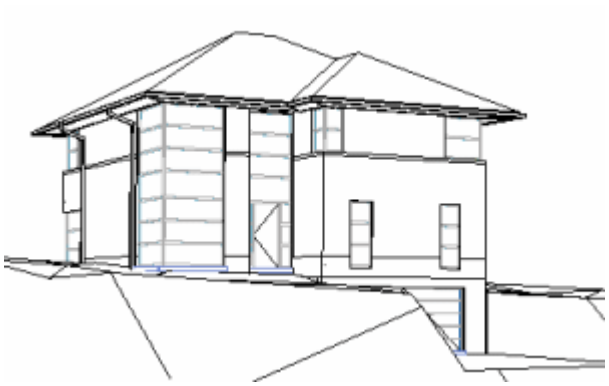
Go to top

Define a View

The Define View command dynamically defines parallel perspective or visual perspective views of your 3D models.



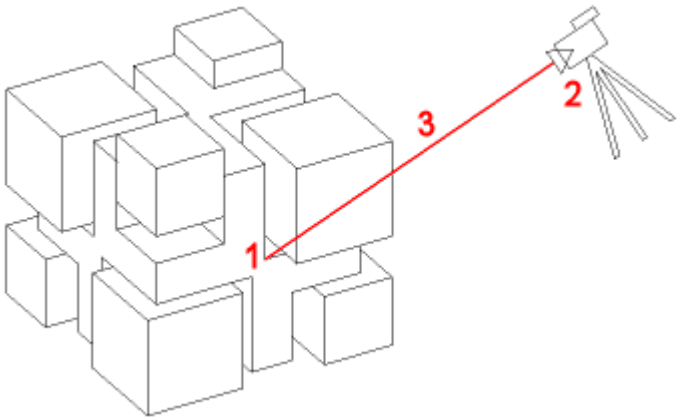
Parallel Perspective View



Visual Perspective View

The following parameters are used in the Define View procedure:

- *Target Point*: Defines the center of the view (1)
- *Camera Point*: Defines the view point and view direction (2).
- *Distance to Target*: Distance between the camera position an the target point (3).



Defining a view

1. Do one of the following:

- Click the Define View tool button ( ) on the *View* toolbar.
- Choose Define View in the *View* menu.

- Type *DV* or *dview* in the command bar.

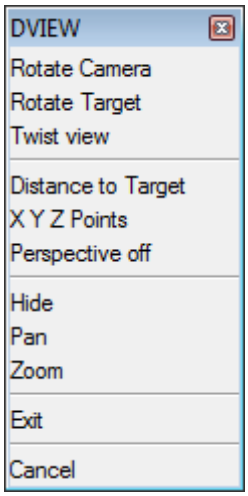
The command bar reads: Enter for all entities/<Select entities to see in preview>:

2. Press Enter to include all entities in the preview.

The command bar reads: Dview:

CAmera/TArget/TWist/Distance/POints/Off/Hide/PA/Zoom/<eXit>:

A prompt box displays:



3. Do one of the following:

- Choose *XYZ Points* in the prompt box.
- Type *PO*, then press Enter.

The command bar reads: Enter target point <current target point>:

4. Specify the target point.

The command bar reads: Enter camera point <current camera point>:

5. Specify the camera point.

The view is rotated accordingly. The perspective type is parallel.

6. (option) To switch to visual perspective view do one of the following:

- Choose *Distance to target* in the prompt box.
- Type *D*, then press Enter.

The command bar reads: New camera-to-target distance <current distance>:

Do one of the following:

- Press Enter to accept the current camera-to-target distance.
- Type a new camera-to-target distance, then press Enter.

7. Choose *Exit* in the prompt box or type *X*, then press Enter to conclude the Define View command.

Command Options

Keyboard	Prompt Box	Description
CA	Rotate Camera	Rotate the *Camera point* about the *Target Point*.
TA	Rotate	Rotate the *Target Point* about the *Camera Point*.

	Target	
TW	Twist View	Rotate the view about the view Z-axis. A positive angle rotates the view counterclockwise.
TA	Distance to Target	Define a new distance-to-target distance. If the view currently is a parallel perspective view, this option switches to visual perspective view.
PO	XYZ Points	Define the *Target Point* and *Camera Point*.
O	Perspective Off	Switch to parallel perspective view.
H	Hide	Hide invisible lines.
PA	Pan	Pan the view.
Z	Zoom	Define the view height in drawing units.
X	Exit	Conclude the Define Zoom command.
Esc	Cancel	Abort the Define Zoom command.

NOTE View tools, such as zoom, pan, real-time motion, are not available in visual perspective views. However, you can use the mouse to manipulate the view.

Named Views

Named Views

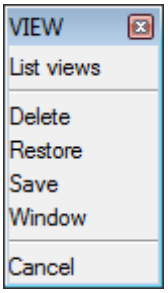
You can save the view displayed in the current window as a named view. Saved views can be restored at any time.

Saving a view

1. Do one of the following:

- Click the Save/Restore View tool button ( ) on the *View* toolbar.
- Choose Save/Restore View in the *View* menu.
- Type *view* in the command bar, then press Enter.

The command bar reads: View: ? to list saved views/Delete/Restore/Save/Window:



2. Do one of the following:

- Choose *Save* in the *View* prompt box.
- Type *S* in the command bar, then press Enter

3. Type a name for the view in the command bar, then press Enter.

The current view is saved.

NOTE If you type a name that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

Command Options

Keyboard	Prompt Box	Description
?	List Views	A list of saved views shows in the Prompt History window. By default all saved views are listed. You can use wildcard characters to limit the number of views. E.g. if you type P* (letter p followed by an asterisk) only views of which the name starts with P are listed.
D	Delete	Delete a named view.
R	Restore	Restore a named view.
S	Save	Save the current view..
W	Window	Save part of the current view. You are prompted to pick two points, which define two opposite corners of a rectangle in order to

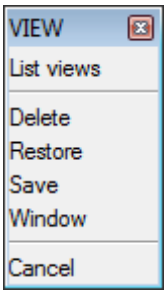
		define the limits of the saved view.
Esc	Cancel	Abort the View command.

Restoring a view

1. Do one of the following:

- Click the Save/Restore View tool button ( ) on the *View* toolbar.
- Choose Save/Restore View in the *View* menu.
- Type *view* in the command bar, then press Enter.

The command bar reads: View: ? to list saved views/Delete/Restore/Save/Window:



2. Do one of the following:

- Choose *Restore* in the *View* prompt box.
- Type *R* in the command bar, then press Enter

3. Type the name of the saved view you want to restore in the command bar, then press Enter.

The saved view is restored.

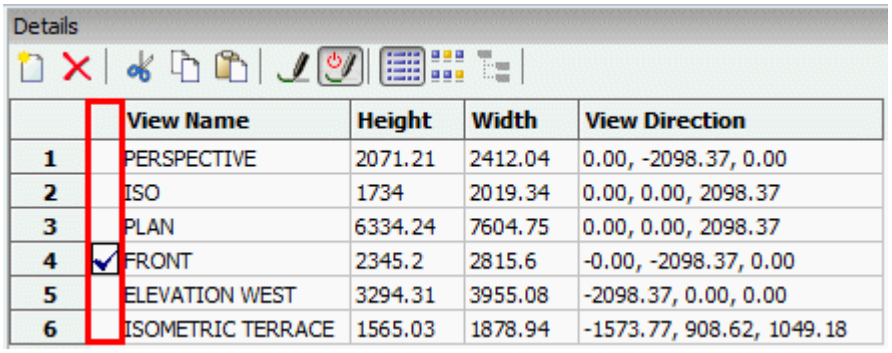
Exploring views

1. Do one of the following:

- Click the Views... tool button ( ) on the *Settings* toolbar.
- Choose Views... in the *Settings* menu.
- Type either *expviews* or *V* in the command bar, then press Enter.

The *Drawing Explorer - Views* window opens.

2. (option) Click the blank tile in front of the *View Name* to restore a saved view.



		View Name	Height	Width	View Direction
1		PERSPECTIVE	2071.21	2412.04	0.00, -2098.37, 0.00
2		ISO	1734	2019.34	0.00, 0.00, 2098.37
3		PLAN	6334.24	7604.75	0.00, 0.00, 2098.37
4	☑	FRONT	2345.2	2815.6	-0.00, -2098.37, 0.00
5		ELEVATION WEST	3294.31	3955.08	-2098.37, 0.00, 0.00
6		ISOMETRIC TERRACE	1565.03	1878.94	-1573.77, 908.62, 1049.18

The currently loaded view is marked.

3. (option) Click the New button ( ) in the *Drawing Explorer* toolbar to save the current view.

4. (option) Click the Delete button ( ) in the *Drawing Explorer* toolbar to delete the selected view.

5. (option) Click the *View Name*, then right click and choose Rename in the context menu. Type a new name for the saved view.

Workspaces

Workspaces

Understanding paper space and model space

When you start a drawing session, your initial working area is called *Model Space*. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the Model tab.

In general model space consists of a single viewport that fills the screen. If needed, you can create additional views called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active click in a viewport to make it the current viewport. You can print the current viewport only. The Viewports command lets you manage your viewports in model space.

Bricscad provides an additional work area, called *Paper Space*. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one *Layout* in which you can have one or more *Layout Viewports*. Such layout viewports are to be considered as a view window of your drawing in model space. Layout viewports can be placed anywere in a paperspace layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The Mview command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

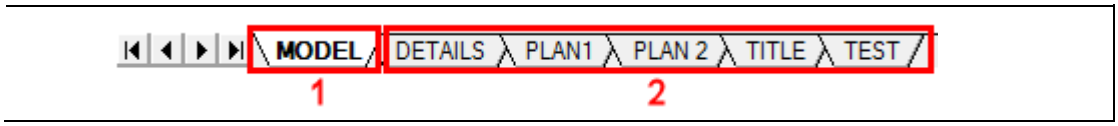
- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

Switching between workspaces

- To switch between Model Space and Paper Space Layouts use the Model and Layout tabs at the bottom of the drawing window.
- When working in a layout, use the Mspace and Pspace commands to toggle between paper space and model space.

Using the model and layout tabs

1. (option) To open model space, click the *Model* tab (1) at the bottom of the drawing window.
2. (option) To open a paper space layout, click the corresponding *Layout* tab (2) at the bottom of the drawing window.



NOTE A drawing contains at least one layout, which is named *Layout1* by default.

Toggle between model space and paper space in a layout

1. (option) When in *model space* (Model Space, with floating viewports) do one of the following to switch to *paper space*:

- Type either *PSPACE* or *PS*, then press Enter
- Double click outside a viewport.

2. (option) When in *paper space* do one of the following to switch to *model space* (Model Space, with floating viewports):

- Type either *MSPACE* or *MS*, then press Enter.
- Double click inside a viewport.

NOTE *Mspace* and *Pspace* commands are available when working in a layout only.

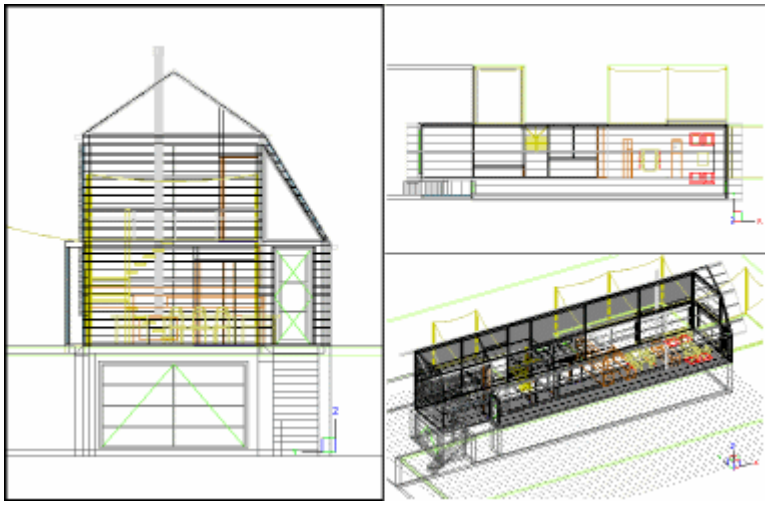
Model Space Viewports

Model Space Viewports

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active click in a viewport to make it the current viewport. You can print the current viewport only.

The following settings can be defined differently for each viewport:

- grid display and snap
- coordinate system: WCS or UCS



Model space divided in 3 viewports

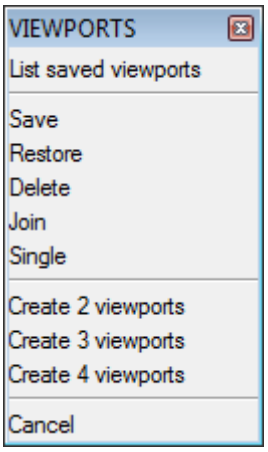
Creating viewports in model space

1. Do one of the following:

- Click the Viewports tool button ( ) on the *Views* toolbar.
- Choose Viewports in the *Views* menu.
- Type *viewports* in the command bar.

  The command bar reads: Viewports: ? to list/Save/Restore/Delete/SIngle/Join/2/3/4/<3>:

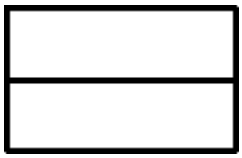
  A prompt box opens.



2. (option) To create 2 viewports, do one of the following:

   - Choose *Create 2 viewports* in the prompt box.
   - Type *2* in the command bar, then press Enter.

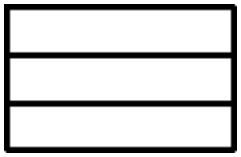
   The command bar reads. Two viewports: Horizontal/<Vertical>:

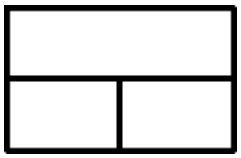
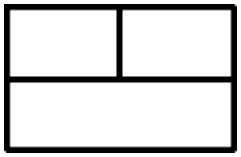
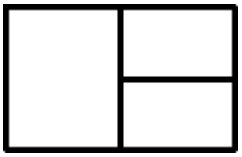
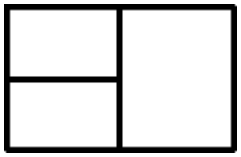
Option	Result
Horizontal	
Vertical	

3. (option) To create 3 viewports, do one of the following:

   - Choose *Create 3 viewports* in the prompt box.
   - Type *3* in the command bar, then press Enter.

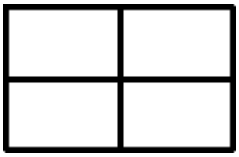
   The command bar reads. Three viewports:
   Horizontal/Vertical/Above/Below/Left/<Right>:

Option	Result
Horizontal	
Vertical	

Above	
Below	
Left	
Right	

4. (option) To create 4 viewports, do one of the following:

- Choose *Create 4 viewports* in the prompt box.
- Type *4* in the command bar, then press Enter.

Option	Result
4	

Command Options

Keyboard	Prompt Box	Description
?	List saved viewports	A list of saved viewport configurations shows in the Prompt History window. By default all saved configurations are listed. You can use wildcard characters to limit the number of configurations . E.g. if you type P* (letter p followed by an asterisk) only configurations of which the name starts with P are listed.
S	Save	Save the current viewport configuration. You are prompted to type a name in the command bar.
R	Restore	Restore a saved viewport configuration. You are prompted to type the name of a saved viewport configuration in the command bar.
D	Delete	Delete a saved viewport configuration. You are prompted to type the name of a saved viewport configuration in the command bar.

J	Join	Join two adjacent viewports.
SI	Single	Restore a single viewport.
2	Create 2 viewports	Divide the current viewport vertically or horizontally.
3	Create 3 viewports	Divide the current viewport into 3 viewports.
4	Create 4 viewports	Divide the current viewport into 4 viewports.
Esc	Cancel	Abort the Viewports command.

Drawing in multiple viewports

1. Click in a viewport to make it current.
   The border of the current viewport highlights.
2. Draw the entities in the current viewport.
3. (option) Repeat step 1 to make a different viewport current.

NOTES

- The crosshairs show in the current viewport only.
- You cannot draw linear entities such as lines, polylines and splines from one viewport to another.
- You can start the Copy or Move command in current window, and finish the operation in a different viewport.

Joining adjacent viewports

1. Launch the Viewports command.
2. Select the *Join* command option.
   The command bar reads: Select inside dominant viewport <Current>:
3. Click in the dominant viewport.
   The border of the selected viewport highlights.
4. Click again in the dominant viewport to confirm.
   The command bar reads. Select inside viewport to join:
5. Click in the viewport you want to join.
   The border of the selected viewport highlights.
6. Click the viewport to join again to confirm.
   The two selected viewports are joined.

NOTE You can join viewports which share an edge of equal length only.

Paper space viewports

Paper space viewports

In a layout you can create multiple viewports each of which display a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

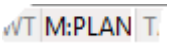
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space with floating viewports (see Toggle between model space and paper space), click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can even turn off the display of the content of a viewport.

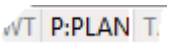
To preserve the scaling of a paper space viewport you can lock the display. It is no longer possible to zoom or pan in a viewport of which the display is locked.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

When you are working in a layout, either *Model Space* or *Paper Space* is your current workspace. The *Workspace* field in the Status Bar indicates which workspace is current: *M:<Layout Name>* indicates you are working in *Model Space*, while *P:<Layout Name>* means *Paper Space* is the current workspace.



Model Space, with floating viewports of layout PLAN



Paper Space of layout PLAN

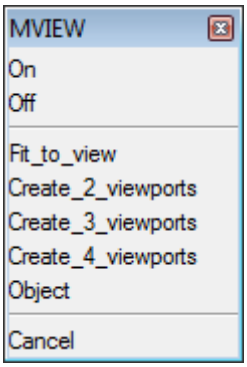
Creating viewports in a layout

1. Click the appropriate layout tab at the bottom of the drawing window.
2. Do one of the following:

- Click the Paper Space Views tool button () on the *Views* toolbar.
- Choose Paper Space Views in the *Views* menu.
- Type *mview* at the command prompt, then press Enter.

The command bar reads: Viewports. ON/OFF/Fit/2/3/4/<First corner>:

A prompt box opens:



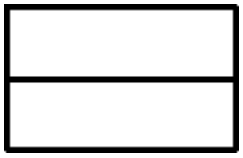
3. (option) To add 1 viewport. do one of the following:

   - Click to specify the first corner of the viewport, then specify the opposite corner.
     A single viewport which fits in the specified rectangle is created.
   - Choose *Fit to view* in the prompt box or type *F*, then press Enter.
     A single viewport which fits in the current drawing display window is created.

4. (option) To create 2 viewports, do one of the following:

   - Choose *Create 2 viewports* in the prompt box.
   - Type *2* in the command bar, then press Enter.

   The command bar reads. Two viewports: Horizontal/<Vertical>:

Option	Result
Horizontal	
Vertical	

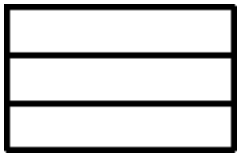
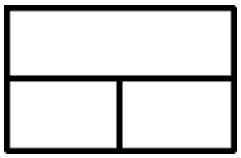
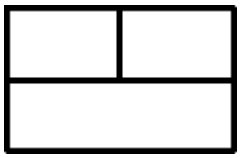
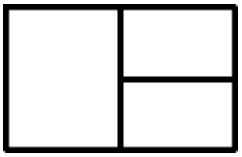
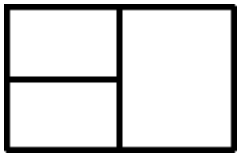
   The command bar reads: Fit to screen/<First corner of bounding rectangle>:

   - Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
     Two viewports which fit in the specified bounding rectangle are created.
   - Choose *Fit to view* in the prompt box or type *F*, then press Enter.
     Two viewports which fit in the current drawing display window are created.

5. (option) To create 3 viewports. do one of the following:

   - Choose *Create 3 viewports* in the prompt box.
   - Type *3* in the command bar, then press Enter.

   The command bar reads. Three viewports:
   Horizontal/Vertical/Above/Below/Left/<Right>:

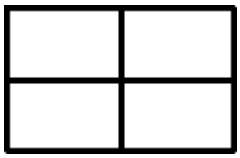
Option	Result
Horizontal	
Vertical	
Above	
Below	
Left	
Right	

The command bar reads: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
  Three viewports which fit in the specified bounding rectangle are created.
- Choose *Fit to view* in the prompt box or type *F*, then press Enter.
  Three viewports which fit in the current drawing display window are created.

6. (option) To create 4 viewports. do one of the following:

- Choose *Create 4 viewports* in the prompt box.
- Type *4* in the command bar, then press Enter.

Option	Result
Create 4 viewports	

The command bar reads: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.

  Four viewports which fit in the specified bounding rectangle are created.

- Choose *Fit to view* in the prompt box or type *F*, then press Enter.

  Four viewports which fit in the current drawing display window are created.

7. (option) To create a non-rectangular (clipped) viewport. do one of the following:

   - Choose *Object* in the prompt box.
   - Type *O* in the command bar, then press Enter.

   The command bar reads: Select Object to clip viewport.

   Select a closed polyline or a circle in the layout.

NOTES

- Make sure no other viewports display when choosing the *Fit to view* option because they will be hidden behind the newly created viewport.
- Each newly created viewport shows all entities on the layers which are currently visible in model space.
- Since a new viewport is created on the current layer it is recommended to use a dedicated layer for viewport entities. Make this layer current when you want to add a new viewport. If you don't want the viewport outlines to be printed, set the *Do Not Plot* property of the viewport layer.
- Only circles and closed polylines can be used to create a clipped viewport. If you want to create an elliptical viewport, use the Polyline Ellipse variable to create a polyline representation of an ellipse.

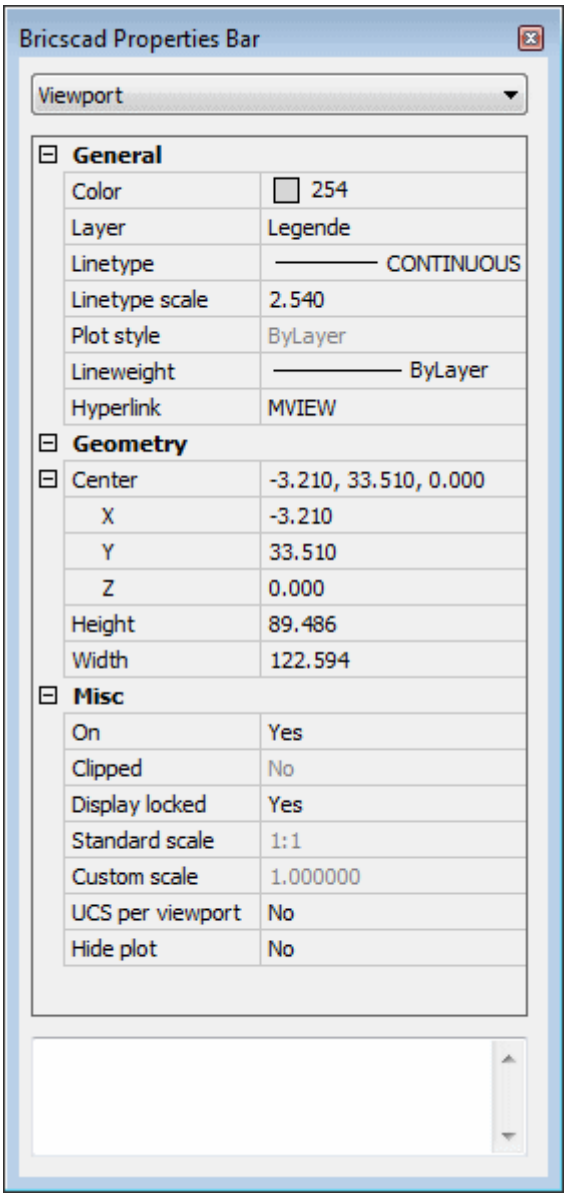
Command Options

Keyboard	Prompt Box	Description
ON	On	Turn the display on of a viewport that was turned off.
OF	Off	Turn the display of a viewport off.
F	Fit to view	Add a single viewport which fits in the current drawing display window.
2	Create 2 viewports	Add two adjacent viewports.
3	Create 3 viewports	Add three adjacent viewports.
4	Create 4 viewports	Add four adjacent viewports.
O	Object	Create a non-rectangular (clipped) viewport.
Esc	Cancel	Abort the Mview command.

Defining viewport properties

1. Click the viewport boundary.

   The viewport properties display in the Bricscad Properties Bar.



2. Click the settings field of a property to modify.

Properties

Name	Description
Center	XYZ coordinates of the center point of the viewport. To specify the center of the viewport graphically, drag the center handle of the viewport.
Height	Height of the viewport in drawing units. To specify the height of the viewport graphically, drag the top or bottom handle of the viewport.
Width	Width of the viewport in drawing units. To specify the width of the viewport graphically, drag the left or right handle of the viewport.
On	Controls the display of the content of the viewport.
Clipped	(not yet implemented)
Display	Locks the scaling of the viewport content to preserve the scaling factor.

locked	
Standard scale	Lets you choose a standard scaling factor.
Custom scale	Lets you define the scaling factor in a decimal format. Displays the current scaling factor.
UCS per viewport	If Yes, lets you define a UCS for this viewport.
Hide plot	Controls whether the viewport boundary is plotted or not.

Navigating in a paper space layout

View manipulation commands, such as zoom, pan and view manipulation using the mouse, act slightly different in a paper space layout compared to model space (with tiled viewports).

If you are working in paper space (no viewport selected), view manipulation commands act on the complete paper space layout.

If you are working in a paper space viewport (model space with floating viewports), view manipulation commands act on the active viewport only. Except if the the display of the current paper space viewport is locked, then the view manipulation commands act on the complete paper space layout.

NOTE It is not possible to rotate the display of a paper space layout. View Rotate commands are available in viewports of which the display is not locked only.

Setting the layer visibility in a paper space viewport

1. Do one of the following:

- If *Paper Space* is the current workspace: double click inside the viewport to make it the current viewport.
- If *Model Space with floating viewports* is the current workspace: click inside the viewport to make it the current viewport.

2. Do one of the following.

- Click the Layers... tool button on the *Settings* toolbar.
- Choose Layers... in the *Settings* menu.
- Type *explayers* in the command bar, then press Enter.

The *Drawing Explorer - Layers* window opens.

3. In the Curr. VP (Current Viewport) column, click the layer(s) you want to freeze.

The thawed icon () is replaced by the frozen icon ().

(option) Click the icon again to thaw a frozen layer.

4. Close the *Drawing Explorer - Layers* window.

5. (option) Repeat steps 1 through 4 to set the layer visibility in another paper space viewport.

Setting the scale of a paper space viewport

1. Switch to *model space* (Model Space, with floating viewports).
   (See Toggle between model space and paper space)
2. Click the viewport border.
   The viewport properties display in the Bricscad Properties Bar.
3. Do one of the following:

- Choose a scale in the *Standard scale* list.
- Type a scaling factor in the *Custom scale* field.

The display of the viewport is scaled with respect to the layout size.

4. If necessary, adjust the viewport borders to the new scale.
5. Set the *Display Locked* property to *Yes*.

NOTE In a locked viewport it is impossible to:

- zoom or pan
- modify the scale

Layouts

Layouts

In Bricscad, you can create multiple layouts for a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. Viewports can display all or part of the drawing's model space entities. Each drawing can contain up to 255 layouts.

To add new layouts you can either define it yourself or your can import a layout from another drawing (template (.dwt) file, drawing (.dwg) file, or drawing interchange (.dxf) file)

Creating a new layout

1. Do one of the following:

- Click the New Layout tool button () on the *Layout* toolbar.
- Choose *Layout* > *New Layout* in the *Insert* menu.
- Type *layout* in the command bar, press Enter, then type *N* and press Enter.

The command bar reads: Enter a layout name. <Layout1>:

2. Do one of the following:

- Type a unique name for the layout, then press Enter.
  The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.
- Right click or press Enter to accept the default name (e.g. Layout1).

The new layout tab is added.

3. Create at least one viewport. (See Creating viewports in a layout for more information.)

Copying layouts

1. Right click the layout tab you want to copy.
   A context menu displays.
2. Choose *Copy* in the context menu.
   The command bar reads: Enter new layout name:
3. Type a new unique name in the command bar, then press Enter.
   A copy of the selected layout is added.

Importing layouts

1. Right click the *Model* tab or one of the layout tabs.
   A context menu displays.
2. Choose *From Template...* in the context menu.
   The *Select Template From File* window opens.
3. Browse to the drawing that you want to import layouts from.
4. Click the *Open* button on the *Select Template From File* window.
   The *Insert Layout(s)* window opens.
5. Select the layout(s) you want to import.
   Press and hold the Ctrl key to select multiple layouts.
6. Click the OK button on the *Insert Layout(s)* window.
   The selected layouts are imported.

Renaming layouts

1. Right click the layout tab you want to rename.
   A context menu displays.
2. Choose *Rename* in the context menu.
   The *Rename Layout* dialog opens.
3. Type a new unique name in the *Name* field of the *Rename Layout* dialog.
4. Click the OK button on the *Rename Layout* dialog.

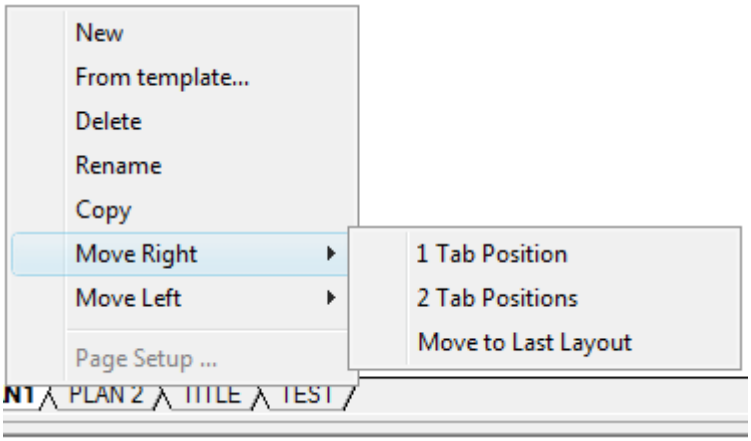
Arranging layout tabs

1. Right click the layout tab you want to move.
   A context menu displays.
2. Do one of the following:

- Choose *Move Right*.
- Choose *Move Left*.

3. To move the selected layout tab do one of the following:

- Select the number of tab positions.
- Choose *Move to Last Layout* or *Move to First Layout*.



Deleting a layout

1. Right click the layout tab you want to move.
   A context menu displays.
2. Choose *Delete* in the context menu.
   The selected layout is deleted.

Drawing Entities

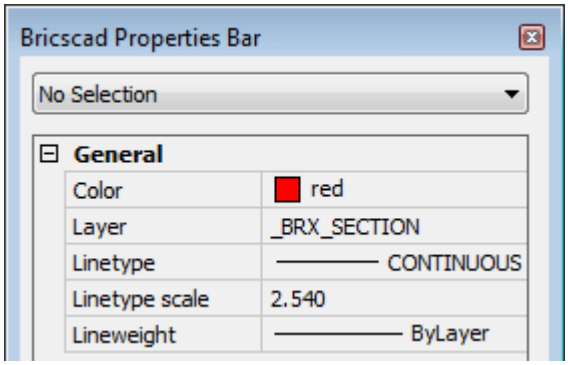
Entity Creation Settings

Overview

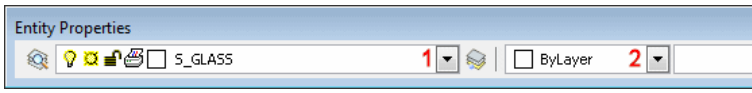
The following settings control the display and/or creation of entities.

Name	Command	Description
Fill mode	FILL FILLMODE	Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in.
Current Entity Color	CECOLOR	Sets the color of new entities.
Lineweight Display	LWDISPLAY	Controls whether lineweights display on the screen.
Default Lineweight	LWDEFAULT	Defines the default lineweight.
Lineweight Units	LWUNITS	Defines whether lineweights are expressed in millimeters or inches.
Current Entity Lineweight	CELWEIGHT	Sets the lineweight for new entities.
Current Entity Linetype Scale	CELTSCALE	Sets the linetype scaling factor for new entities.
Linetype Scale	LTSCALE	Sets the linetype scaling factor for all entities in the drawing (global linetype scale).
Current Entity Linetype	CELTYPE	Sets the linetype for new entities.
Current Layer	CLAYER	Sets the layer for new entities.

The current properties display in the *Bricascad Properties Bar* and in the *Entity Properties* toolbar.



Bricscad Properties Bar

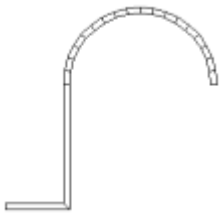
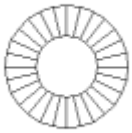
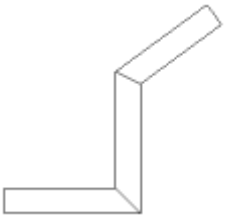
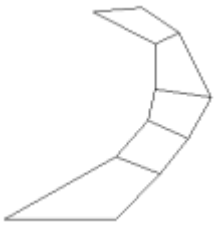


Entity Properties toolbar: 1. Layer 2. Color 3. Linety

Fill Mode

Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in. If Fill Mode is off, all filled entities, display and print as outlines.

You can reduce the time it takes to display or print a drawing by turning off the display of solid fill.

	Polyline	Donut	Trace	Plane (Solid)
Fill On				
Fill Off				

Setting Fill Mode

1. To toggle *Fill Mode* on / off do one of the following:

- Choose Fill in the *Settings* Menu.
- Click the Fill tool button () on the *Settings* toolbar.
- Type *FILL* or *FILLMODE* in the command bar, then type T + Enter.

2. To regenerate the viewport do one of the following:

- Choose Regen in the View menu.
- Click the Regen tool button () on the *View* toolbar.
- Type *REGEN* in the command bar, then press Enter.

NOTE If multiple viewports are open, click the Regen All tool button () on the *View* toolbar to regenerate all open viewports simultaneously.

Checking the Fill Mode setting

To check whether Fill Mode is currently on or off, do one of the following:

- Type *FILL* or *FILLMODE* in the command bar.
  The current status of *Fill Mode* is shown between arrow brackets.
- Check the Fill tool button on the *Settings* toolbar.

If the Fill tool button is pressed, *Fill Mode* is on:

If the Fill tool button is depressed, *Fill Mode* is off:

Entity Color

Entity Color

An entity's color determines how it is displayed and how it prints. Entities are created in the current color.

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

Index Colors

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

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

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

Color BYLAYER:

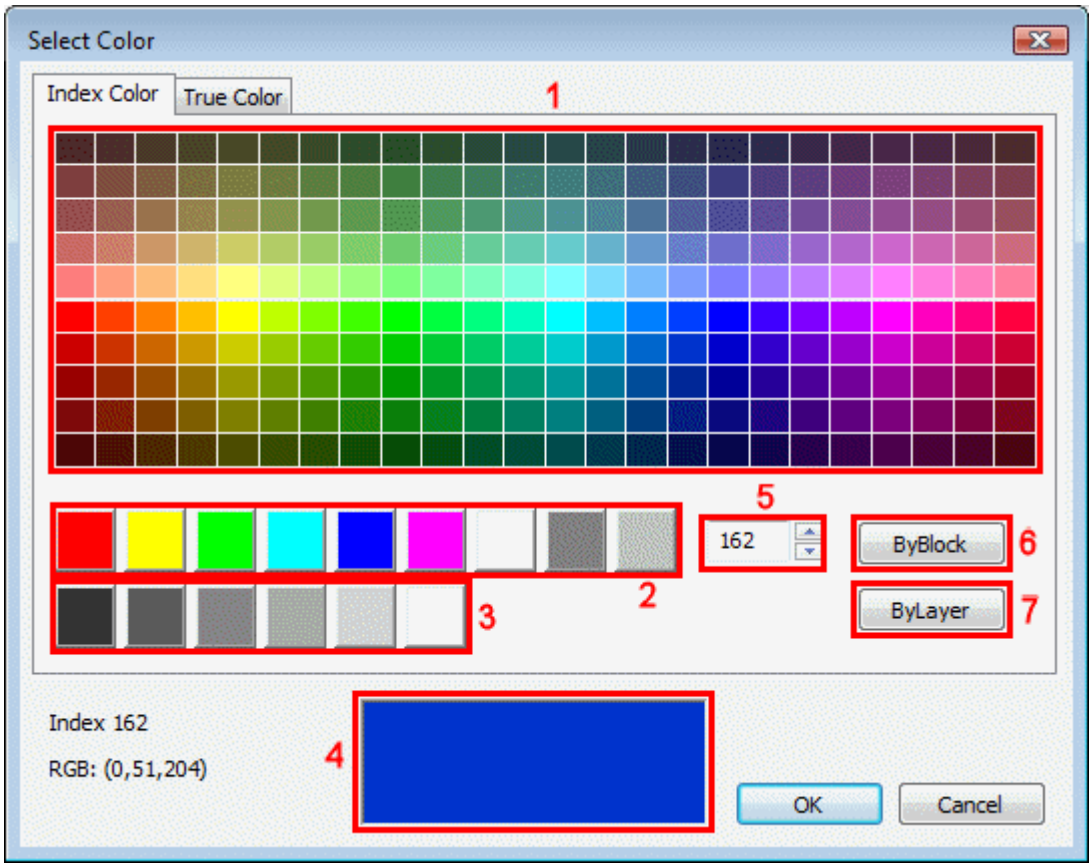
Entities which have a color BYLAYER adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

Color BYBLOCK:

Entities which have a color BYBLOCK are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block.

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

Index color dialog



1. Colors 10 - 249
2. Colors 1 to 9
3. Colors 250 - 255
4. Current Color
5. Color Number field
6. 'Color By Block' button
7. 'Color By Layer' button

True Colors

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

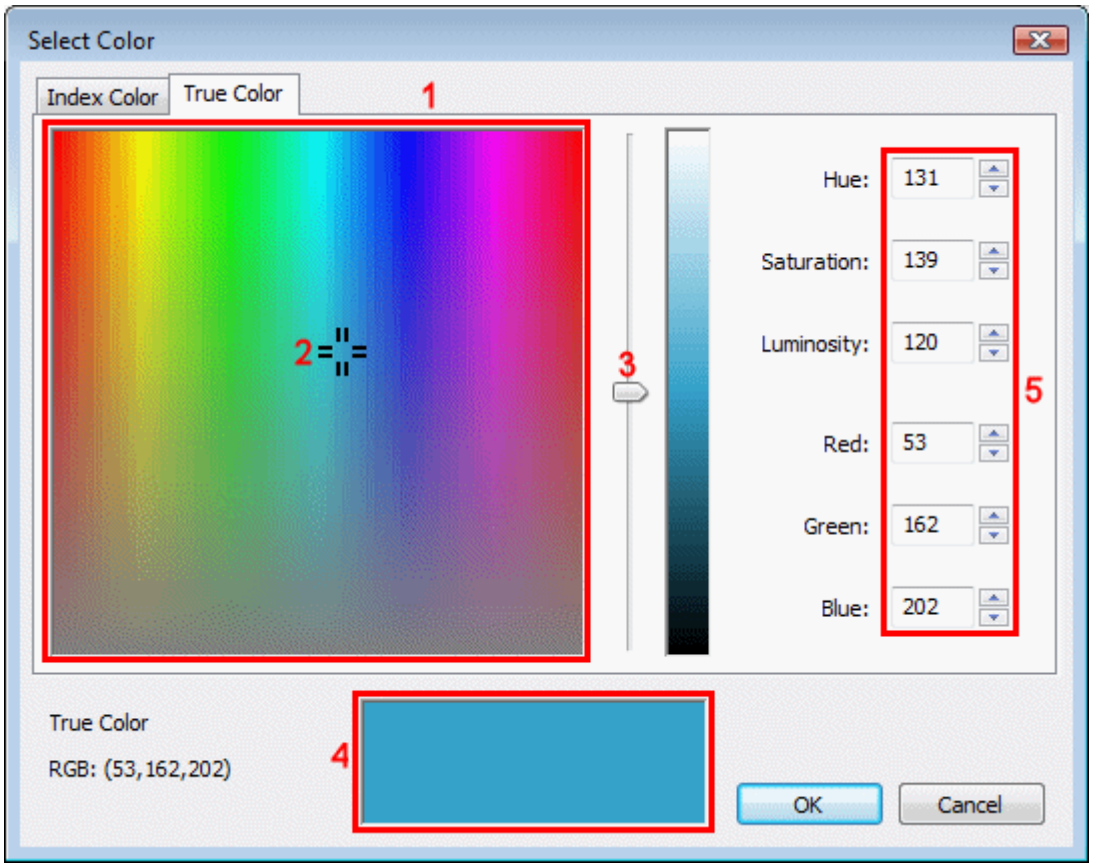
The number of bits used to define a pixel's color shade is its *bit-depth*. True color is sometimes known as *24-bit color*. Some new color display systems offer a 32-bit color mode. The extra byte, called the *alpha channel*, is used for control and special effects information.

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

NOTE In drawings that use *Style Tables* (STB) to set up plotting configurations, entities in a true color will only be printed in this color if the *Color* setting in the STB definition file is set to *Use Object Color.*

In drawings that use *Color Tables* (CTB) to set up plotting configurations, entities in a true color always print in this color.

True color dialog



1. Color Selection pane
2. Color Indicator
3. Luminosity slider
4. Current Color
5. Color Parameter fields

Setting the current entity color

1. To open the *Select Color* dialog, do one of the following:

- In the *Color* field of the *Entity Properties* toolbar, choose *Select Color*.
- Click the Select Color tool button ( ) on the *Settings* toolbar.
- In the Bricscad Properties Bar click *Color*, then choose *Select Color* in the drop down list.
- Double click the *Color Field* in the Status Bar.
- Type SETCOLOR in the command bar.

2. (option) To select one of the *index colors*, do one of the following:

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

3. (option) To define a true color click the *True Color* tab on the *Select Color* dialog, then do one of the following:

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

4. Click the *OK* button to confirm.

Choosing a named color

To set the entity color to one of the named colors, do one of the following:

- Select the color in the *Color* field of the *Entity Properties* toolbar.
- Select the color on the *Color* flyout of the *Settings* toolbar.
- In the Bricscad Properties Bar click *Color*, then select the color in the drop down list.
- Right click the *Color Field* in the Status Bar, then select the color in the context menu.

Lineweight

Lineweight

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

The following lineweights are available: BYLAYER, BYBLOCK, DEFAULT and many additional lineweights in millimeters or inches, depending on the *Lineweight Units* (LWUNITS) setting..

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

New entities are drawn using the current lineweight.

Lineweight BYLAYER:

When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity lineweight is determined by the current layer. When you assign BYLAYER, changing a layer's lineweight changes the lineweight of all such entities on that layer.

Lineweight BYBLOCK:

Entities created using lineweight BYBLOCK, are drawn using the DEFAULT lineweight until include them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

DEFAULT lineweight:

The default lineweight is saved in the *Default Lineweight* setting (LWDEFAULT). Entities created using the default lineweight will be adjusted if the *Default Lineweight* setting is redefined.

NOTE For entities using an index color, lineweights will be used for printing only if the *Color Table* (CTB) or *Plot Style* (STB) is defined as 'Use object lineweight'. Otherwise the line weight is defined by the objects color or plot style.

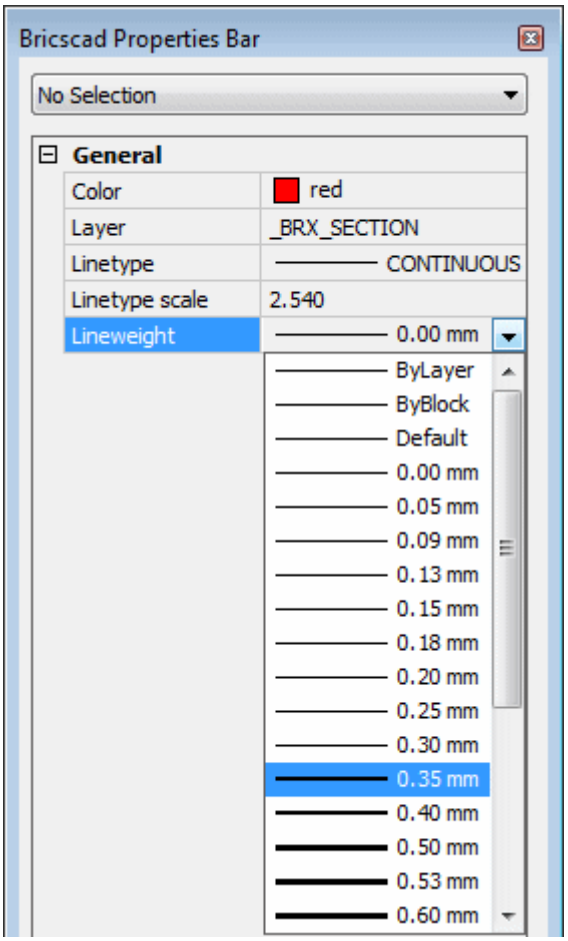
Entities which are created in a true color always use the object line weight when printed.

Setting the current lineweight

1. (option) On the Bricscad Properties Bar, select *Lineweight*.

   Make sure, no entity is selected.

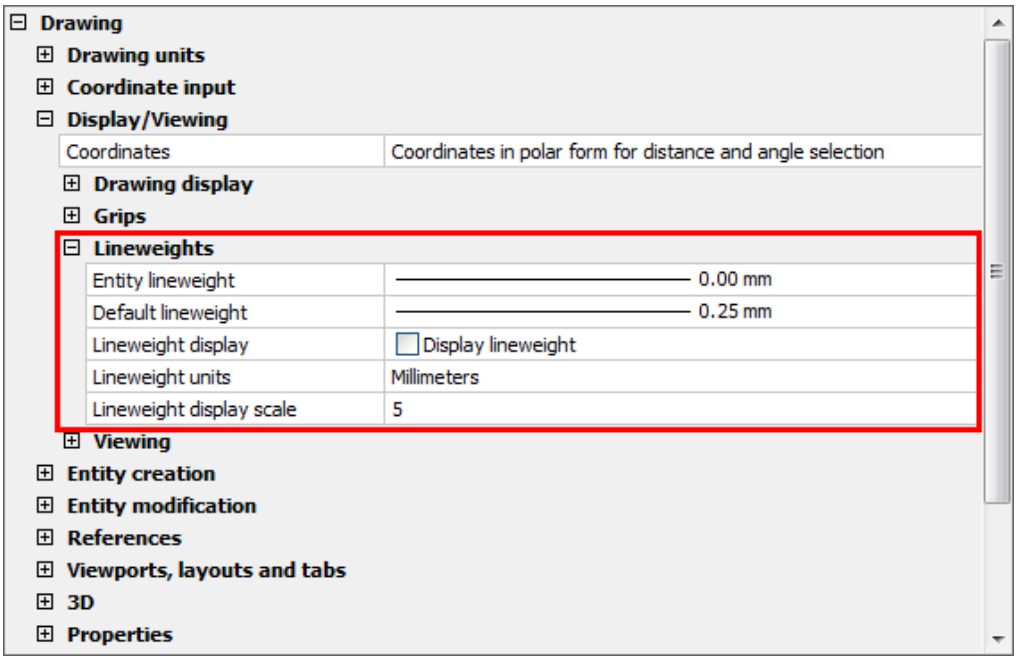
Choose a lineweight in the drop down list.



2. (option) On the *Entity Properties* toolbar, click the down arrow next to the *Lineweight* field, then choose a lineweight in the drop down list.

Defining the lineweight settings

1. Choose *Settings* in the *Settings* menu.
2. Select the *Drawing settings* category ( )
3. Expand the *Display/Viewing* settings sub-category.
4. Expand the *Lineweights* settings group.



5. (option) Select the *Default Lineweight* setting (LWDEFAULT), then select a lineweight in the drop down list.
6. (option) Select the *Lineweight Display* setting (LWDISPLAY), then click the checkbox to toggle the display of lineweights.
7. (option) Select the *Lineweight Units* setting (LWUNITS), then choose *Millimeters* or *Inches*.
8. Close the *Settings* dialog.

NOTE You can toggle the display of lineweights on/off in the Status Bar also.

Linetype

Entity Linetype

You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER and BYBLOCK. Your drawing may also contain an unlimited number of additional linetypes. Linetype definitions are saved in the drawing. New linetypes are either imported from other drawings or loaded from a linetype file (e.g. iso.lin or default.lin).

New entities are drawn using the current lineweight.

Linetype BYLAYER:

Entities which have a linetype BYLAYER adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.

Linetype BYBLOCK:

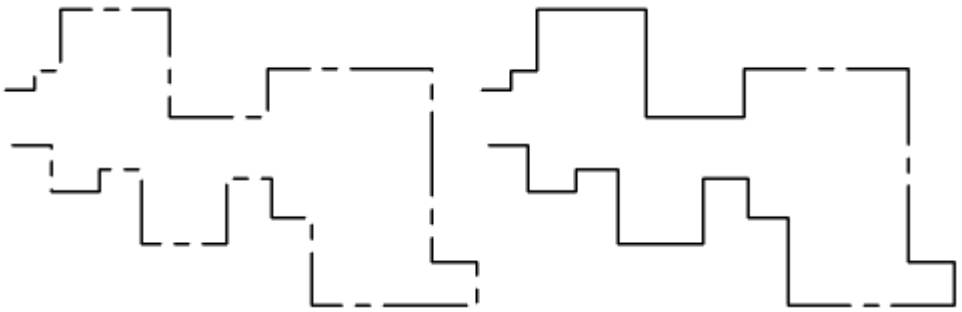
Entities created using linetype BYBLOCK, are drawn as continuous lines until you include them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

Linetype SCALE

Non-continuous linetypes consist of lines, gaps and dots put together in a variety of patterns. In complex linetypes also text or shapes can be included. In the linetype definition the length of the lines and gaps are defined in drawing units. To display the linetype correctly it needs to be scaled to match the dimensions of your drawing. The linetype scaling factor is defined by the *Linetype Scale* setting (LTSCALE). Bricscad will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

Because linetypes are affected by scale, paper space becomes a problem. A linetype scale that looks fine in model space is possibly going to look wrong in paper space. This problem is solved by the *Paperspace Linetype Scale* setting (PSLTSCALE). When set to 0 or Off, PSLTSCALE scales linetypes the same in model space and in layouts, when set to 1or On, linetypes in layouts are drawn to the same scale as the viewport's scale.

Linetypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. The *Polyline Generation* setting (PLINEGEN) solves this problem: when set to 1, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex.



The same polyline drawn with PLINEGEN on (left) and off (right).

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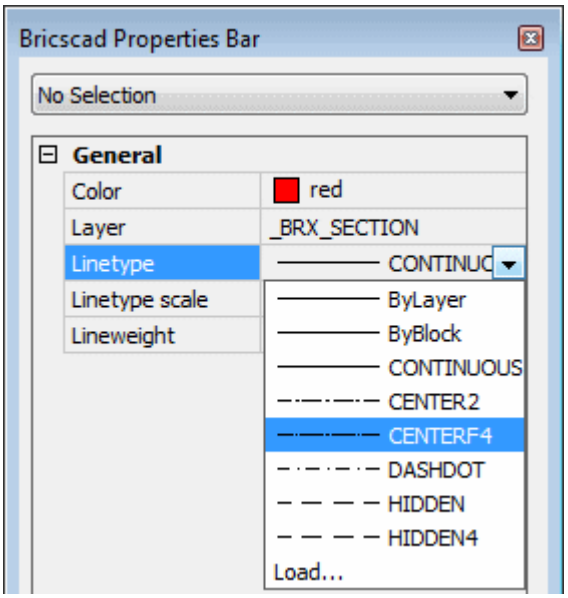
Setting the current linetype

1. (option) On the Bricscad Properties Bar, select *Linetype*.
   Make sure, no entity is selected.

   Choose a linetype in the drop down list.



2. (option) On the *Entity Properties* toolbar, click the down arrow next to the *Linetype* field, then choose a linetype in the drop down list.

3. (option) Right click on the *Linetype* field in the Status Bar, then choose a linetype in the context menu.

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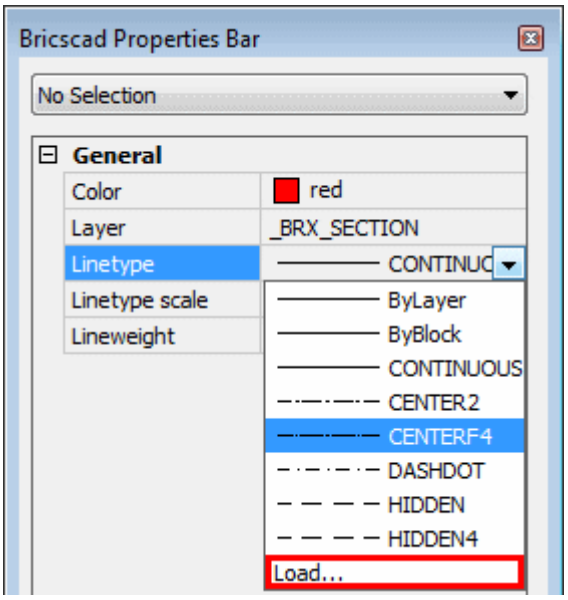
Adding a new linetype

1. On the Bricscad Properties Bar, select *Linetype*.
   Make sure, no entity is selected.
2. Choose *Load...* in the drop down list.



The *Linetypes* dialog opens.

3. (option) Click the *File* button on the *Linetypes* dialog to load a different linetype file (*.lin).

4. Select a linetype, then click the *OK* button.

The linetype is loaded.

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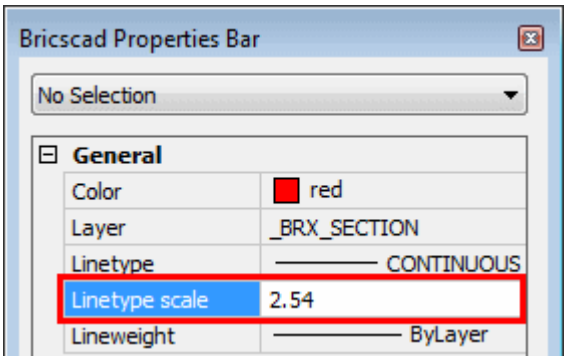
Setting the Entity Linetype Scale

1. On the Bricscad Properties Bar, select *Linetype scale*.
   Make sure, no entity is selected.

2. Type a new value in the Linetype scale field..



NOTE Since most linetypes are defined in inches, it is recommended to set the linetype scale equal to 1 inch, expressed in your drawing units. If you use mm as your drawing units, for cm set the linetype scale to 25.4 and use 0.0254 for meters.

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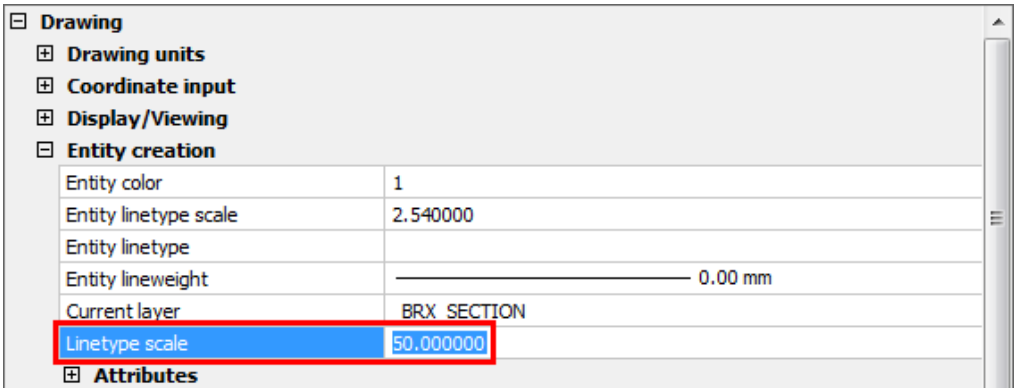
Setting the Global Linetype Scale

1. Choose *Settings* in the *Settings* menu.
2. Select the *Drawing settings* category ( )
3. Expand the *Entity Creation* settings sub-category.
4. Select the *Global Linetype Scale* setting (LTSCALE).



5. Type a new value in the *Global Linetype Scale* field, then press Enter.

   The drawing is regenerated.
6. Close the *Settings* dialog.

NOTE The *Global Linetype Scale* must be set with respect to the plot scale. E.g. if you want to plot your drawing at 1/50, set the LTSCALE setting to 50. Remember to set the *Paperspace Linetype Scale* setting (PSLTSCALE) *Off* if you want linetypes to be scaled equally in model space and paper space.

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Current Layer

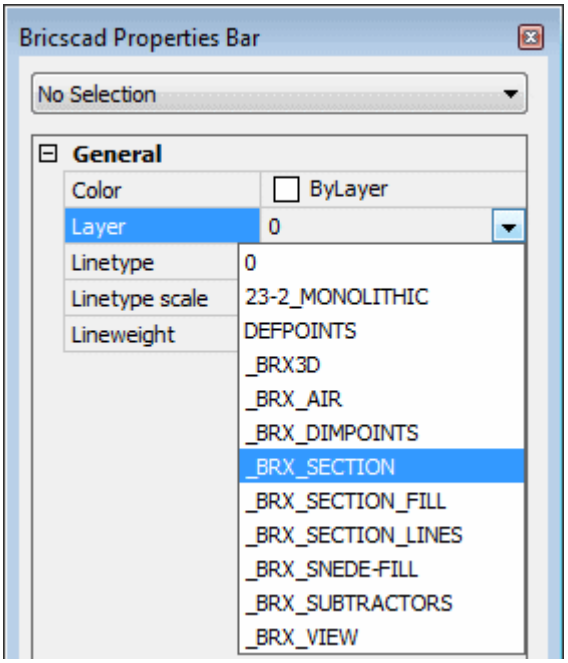
When you create new entities, they are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

Setting the current layer

1. (option) On the Bricscad Properties Bar, select *Layer*.

   Make sure, no entity is selected.

   Choose a layer in the drop down list.



2. (option) On the *Entity Properties* toolbar, click the down arrow next to the *Layer* field, then choose a layer in the drop down list.

3. (option) Right click on the *Layer* field in the Status Bar, then choose a layer in the context menu.

Drawing 2D Entities

Overview

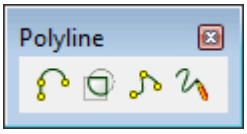
All tools to create 2D entities can be found either on the *Draw 2D* toolbar or in the *Draw* menu.

Toolbars

Draw 2D flyouts:

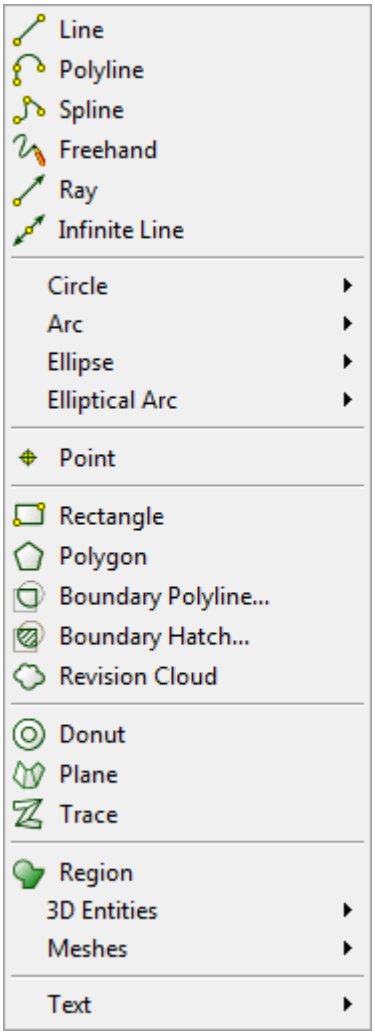
1. Line - Ray - Infinite Line

2. Polyline - Boundary Polyline - Spline - Freehand



3. Rectangle - Polygon (3 methods) - Trace - Donut - Plane - Revision Cloud

Menu



Icon	Tool name	Keyboard	Description
	Line	line	Creates single lines or a series of connected lines.
	Ray	ray	Creates a line that starts at a point and extends to infinity.
	Infinite Line	infline	Creates a line through a given point, oriented at a specified angle and extending to infinity in both directions.
	Polyline	pline	Creates a single open or closed entity, composed of lines and/or arcs.

	Boundary Polyline	bpoly boundary	Calculates a closed polyline from a boundary set.
	Spline	spline	Creates open or closed curved lines.
	Freehand	freehand	Creates lines as if you were sketching, using a pencil.
	Circle	circle	Creates circles.
	Arc	arc	Creates circular arcs.
	Ellipse	ellipse	Creates ellipses
	Elliptical Arc	ellipse	Creates elliptical arcs
	Rectangle	rectangle	Creates Rectangles
	Polygon	polygon	Creates equal sided polygons.
	Trace	trace	Creates traces.
	Donut	donut	Creates donuts.
	Plane	plane	Creates planes (solids).
	Revision Cloud	revcloud	Creates revisioun clouds.
	Point	point	Creates points.

Lines

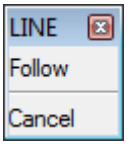
A line consists of two points: a start point and an endpoint. Using the Line command you can draw a series of connected lines, but each line is considered a separate line entity.

Drawing lines

1. Do one of the following

- Click the Line tool button ( ) on the *Draw 2D* toolbar.
- Choose Line in the *Draw* menu.
- Type *line* in the command bar, then press Enter.
- Type *L* in the command bar, then press Enter.

The command bar reads: ENTER to use last point/Follow/<Start of line>:
A prompt box opens:



2. Specify the start point.

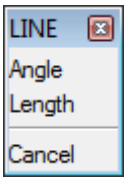
The line displays dynamically.

3. Specify the end point.

The line is drawn and the following line displays dynamically starting in the endpoint of the first line.

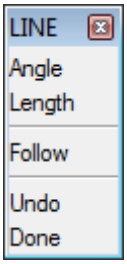
- Using Ortho Mode or Polar Tracking you can type the length of the line in the command bar.
- Using Snap Tracking you can position the end point with respect to the start point and any existing point in the drawing.

The prompt box changes:

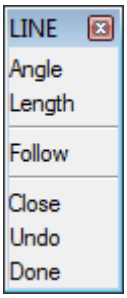


4. (option) Repeat step 3 to draw a series of connected lines.

When the second line is drawn, the options in the prompt box are:



After the third line, the options in the prompt box are:



4. Right click to stop drawing lines.

5. (option) Right click to restart.

Command Options

Keyboard	Prompt Box	Description
Enter		The last point used in a previous drawing tool is used as the start point of the line. Concludes the Line command.
F	Follow	The line is created as an extension of the previously drawn entity. If this was an arc, the line is drawn tangent to the arc, starting at the endpoint of the arc.
A	Angle	Type the angle for the next line. Positive angles are measured counter clockwise from the positive x-axis. The *Length* option follows by default.
L	Length	Type the length for the next line.
C	Close	Connects the end point of the last line to the start point of the first line and concludes the Line command.
U	Undo	Undoes the previous action.
D	Done	Concludes the Line command.

Rays

A ray is a line that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing a ray is to select the start point of the ray, and then specify its direction.

Rays and Infinite Lines or sometimes referred to as construction lines.

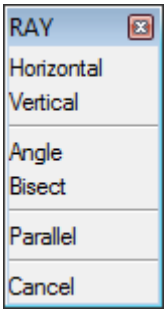
Drawing rays

1. Do one of the following:

- Click the Ray tool button ( ) on the *Draw 2D* toolbar.
- Choose Ray in the *Draw* menu.
- Type *ray* in the command bar, then press Enter.

  The command bar reads: Infinite ray: Bisect/Horizontal/Vertical/Angle/Parallel/<Start of Ray>:

  A prompt box opens:



2. Specify the start point of the ray.

The ray displays dynamically.

3. Specify the direction of the ray.

The next ray displays dynamically.

4. (option) Keep specifying directions to draw a series of rays, starting from the same point.

5. Right click to stop drawing rays.

Command Options

Keyboard	Prompt Box	Description
H	Horizontal	The ray is drawn parallel to the x-axis of the current coordinate system.
V	Vertical	The ray is drawn parallel to the y-axis of the current coordinate system.
A	Angle	The ray is drawn at the specified angle. You can define the angle in one of two ways: • type the angle in the command bar • define the angle by clicking two points in the drawing

B	Bisect	The ray is drawn perpendicular to an existing entity. The startpoint of the ray is the midpoint of the selected entity. You are prompted to choose the side of the selected entity where the ray must be created.
P	Parallel	The ray is drawn parallel to an existing entity. You are prompted to create the ray at a specified distance and side of the existing entity or to draw the ray through a point.
C	Cancel	Concludes the command.

Infinite Lines

An infinite line is a line through a given point, oriented at a specified angle and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing an infinite line is to select a point and then specify its direction.

Rays and Infinite Lines or sometimes referred to as construction lines.

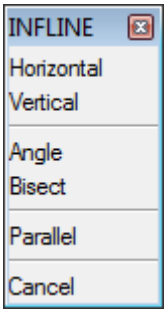
Drawing infinite lines

1. Do one of the following:

- Click the Infinite Line tool button () on the *Draw 2D* toolbar.
- Choose Infinite Line in the *Draw* menu.
- Type *infline* in the command bar, then press Enter.

  The command bar reads: Infinite line: Bisect/Horizontal/Vertical/Angle/Parallel/<Start of Ray>:

  A prompt box opens:



2. Specify the start point of the infinite line.

The infinite line displays dynamically.

3. Specify the direction of the infinite line.

The next infinite line displays dynamically.

4. (option) Keep specifying directions to draw a series of infinite lines, starting from the same point.

5. Right click to stop drawing infinite lines.

6. (option) Right click to restart.

Command Options

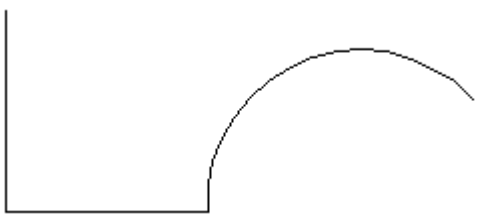
Keyboard	Prompt Box	Description
H	Horizontal	The infinite line is drawn parallel to the x-axis of the current coordinate system.
V	Vertical	The infinite line is drawn parallel to the y-axis of the current coordinate system.
A	Angle	The infinite line is drawn at the specified angle. You can define the angle in one of two ways:

		• type the angle in the command bar • define the angle by clicking two points in the drawing
B	Bisect	The infinite line is drawn perpendicular to an existing entity. The start point of the infinite line is the midpoint of the selected entity. You are prompted to choose the side of the selected entity where the infinite line must be created.
P	Parallel	The infinite line is drawn parallel to an existing entity. You are prompted to create the infinite line at a specified distance and side of the existing entity or to draw the infinite line through a point.
C	Cancel	Concludes the command.

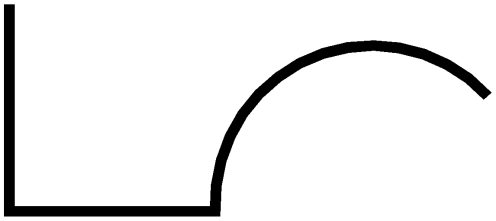
Polylines

Polylines

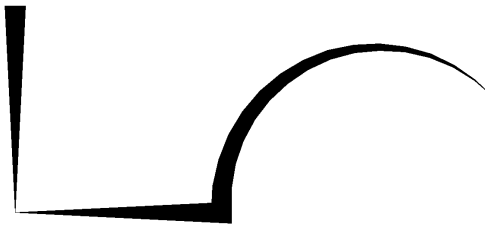
A polyline is an open or closed sequence of connected line and arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment. When a polyline is edited, you can modify the entire polyline or change individual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



zero width polyline



constant width polyline



tapered width polyline

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

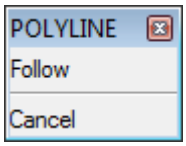
Drawing polylines

1. Do one of the following

- Click the Polyline tool button ( ) on the *Draw 2D* toolbar.
- Choose Polyline in the *Draw* menu.
- Type *pline* in the command bar, then press Enter.
- Type *PL* in the command bar, then press Enter.

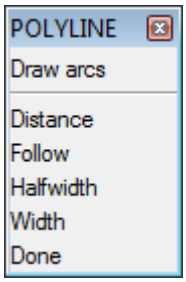
The command bar reads: ENTER to use last point/Follow/<Start of polyline>:

A prompt box opens:



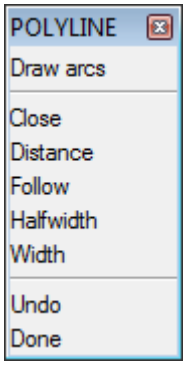
2. Specify the start point of the polyline.

The prompt box changes:



3. Specify the second point of the polyline.

The prompt box changes:

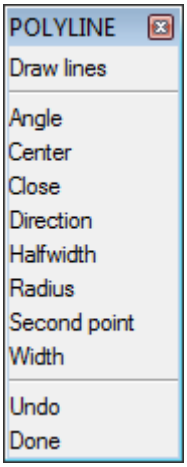


4. (option) Repeat step 3 to add more straight segments.

5. (option) Do one of the following to start drawing arc segments:

- Type *A*, then press Enter.
- Choose *Draw arcs* in the prompt box.

The prompt box changes:



An arc segement displays dynamically. The arc is tangent to the previous line segment.

6. Specify the endpoint of the arc.

7. (option) Repeat step 6 to add more arc segments.

8. (option) Do one of the following to start drawing line segments:

- Type *L*, then press Enter.
- Choose *Draw lines* in the prompt box.

8. Specify the endpoint of the line segment.

9. Right click or choose *Done* in the prompt box to stop.

10. (option) Right click to restart.

Command Options

Keyboard	Prompt Box	Description
Enter		The last point used in a previous drawing tool is used as the start point of the first segment. Concludes the command.
A	Draw arcs	When drawing lines, start drawing arc segments.
A	Angle	When drawing arcs, specify the angle for the arc segment. You are then prompted to specify either the ***E**ndpoint*, ***C**enter* or ***R**adius* of the arc (*).
C	Close	When drawing lines, closes the polyline.
CL	Close	When drawing arcs, closes the polyline.
CE	Center	When drawing arcs, specify the center point of the arc segment. You are then prompted to specify either the ***A**ngle*, ***L**ength* or endpoint of the arc (*).
D	Distance	When drawing lines, specify the length of the next segment. The *Angle* option follows by default.
D	Direction	When drawing arcs, specify the direction for a straight segment. You are prompted to specify the endpoint of the arc. This option allows you to draw line segments when drawing arcs if the endpoint of the arc lies in the specified direction.
F	Follow	The segment is created as an extension of the previously drawn entity or segment. If this was an arc, the segment is drawn tangent to the arc, starting at the endpoint of the arc.
H	Half width	Specify a new current *width* for polyline segments. The *width* will be twice the keyboard entry, expressed in drawing units. You are prompted to specify the *start width* first, then the *end width*. The current width shows between arrow brackets. To accept the current width, press Enter. By default the *end with* equals the *start width*.
L	Draw lines	When drawing arcs, start drawing line segments.
W	Width	Specify a new current *width* for polyline segments. The *width* equals the keyboard entry, expressed in drawing units. You are prompted to specify the *start width* first, then the *end width*. The current width shows between arrow brackets. To accept the current width, press Enter. By default the *end with* equals the *start width*.
R	Radius	When drawing arcs, specify the radius of the arc segment. You are prompted to specify the ***A**ngle* (*) or the endpoint of the arc. If you have chosen to specify the angle, you are prompted to specify the direction of the chord.

S	Second point	When drawing arcs, draws the arc segment through the specified point. This point is NOT a vertex of the polyline.
U	Undo	Removes the last segment.
D	Done	Concludes the command.

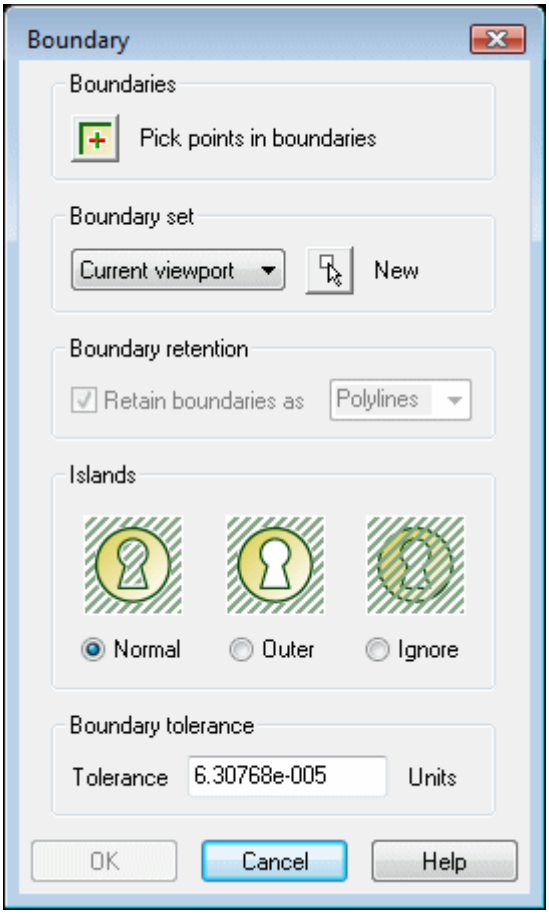
(*) Type the capitalized letter, followed by Enter to choose an option.

Creating boundary polylines

1. Do one of the following

- Click the Boundary Polyline tool button ( ) on the *Draw 2D* toolbar.
- Choose Boundary Polyline in the *Draw* menu.
- Type *bpoly* in the command bar, then press Enter.

The *Boundary* dialog box opens.



2. (option) Click the *New Boundary Set* button ( )

The *Boundary* dialog closes.

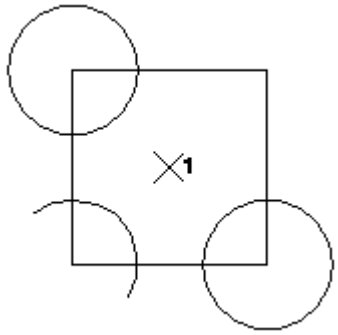
Select the boundary entities in the drawing, then Press Enter.

The *Boundary* dialog opens.

3. Click the *Pick points in boundaries* () button on the *Boundary* dialog box.

The dialog box closes.

4. Click in the area where you want to create the boundary polyline: point 1 in the image below.



5. The boundary polyline displays in dashed lines.

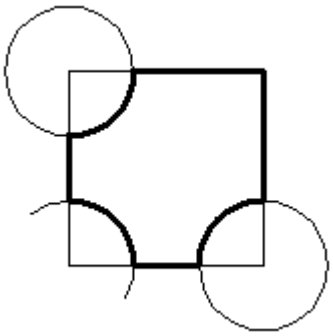
6. (option) Keep picking points to create more boundary polylines.

7. Right click to accept the polyline(s).

The *Boundary* dialog box opens again.

8. Click the *OK* button on the *Boundary* dialog box.

The polyline(s) is (are) created.



NOTES

- If you select the boundary entities before launching the Boundary Polyline tool, you can make the selection set active by pressing the *New Boundary Set* button () in step 2.
- If you type *-boundary* (don't forget the "minus" sign) in the command bar, you are prompted to click the area where you want to create the polyline(s) immediately (no dialog opens) then right click to create the polyline(s).

Splines

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved shapes, which cannot be drawn as a polyline.

Drawing splines

1. Do one of the following

- Click the Spline tool button ( ) on the *Draw 2D* toolbar.
- Choose Spline in the *Draw* menu.
- Type *spline* in the command bar, then press Enter.

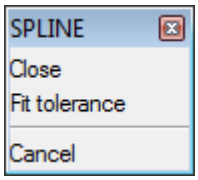
The command bar reads: First point of spline:

2. Specify the first point of the spline.

3. Define the second point of the spline.

The command bar reads: Close/Fit Tolerance/<next point>:

A prompt box displays.

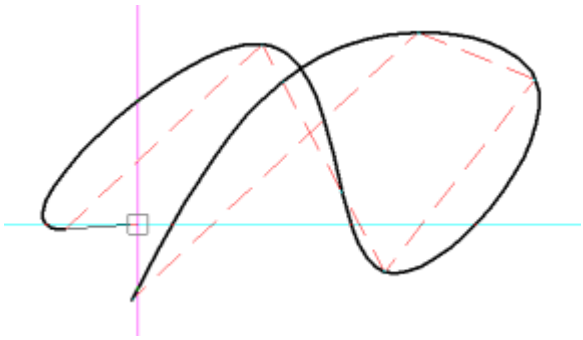


4. (option) Repeat step 3 to define more points.

5. Right click to stop adding more points.

The command bar reads: Select starting tangent point:

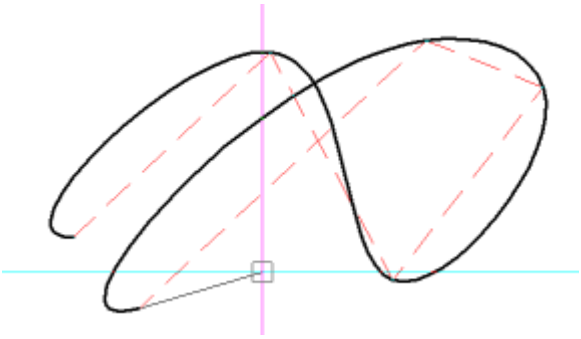
A line that is tangent to the spline displays dynamically from the start point of the spline.



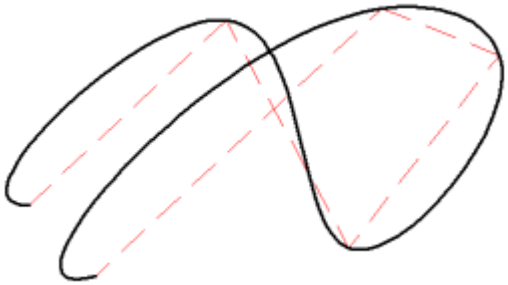
6. Click to define the curve of the spline.

The command bar reads: Select ending tangent point:

A line that is tangent to the spline displays dynamically from the start end of the spline.



7. Click to conclude the spline command.



8. (option) Right click to restart.

NOTE The red dashed lines in the above images represent the polyline through the control points of the spline.

Command Options

Keyboard	Prompt Box	Description
C	Close	Create a closed spline. In this case the spline command concludes after defining the starting tangent point.
F	Fit Tolerance	By default, the spline passes through all of the control points. When you draw a spline, you can change this by specifying the fit tolerance. The fit tolerance value determines how closely the spline fits the set of points you specify. For example, a spline fit tolerance value of 0 (zero) causes the spline to pass through the control points. A value of 0.01 creates a spline that passes through the start and endpoints and within 0.01 units of the intermediate control points.
	Cancel	Ends the spline command without creating the spline.

Freehand

A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline. Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but small segments increase the file size.

After you specify the length of the sketch segments, the crosshairs change to a Pencil tool. A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

Whether a chain of individual lines or a polyline is created by the Freehand tool is controlled through the *Sketch Poly* (skpoly) setting. When the skpoly setting is *On* the Freehand tool creates polylines.

Creating a freehand sketch

1. Do one of the following

- Click the Freehand tool button ( ) on the *Draw 2D* toolbar.
- Choose Freehand in the *Draw* menu.
- Type *freehand* in the command bar, then press Enter.

The command bar reads: Freehand: Length of segments <current length>:

2. Do one of the following to define the length of the segments::

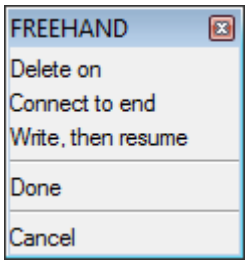
- Press Enter to accept the current length.
- Type a new value in the command bar.
- Define the length graphically by clicking two points.

The new value is saved in the SKETCHINC setting.

3. Click to start sketching (pen down). The crosshairs change to into the Pencil tool ( )

The command bar reads: Press ENTER to end /Pen up/Quit/Delete on /Connect /Straight to cursor /Write to drawing /(Sketching ...):

A prompt box displays:



4. Move the cursor to sketch.

The sketched line displays in light green.

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

6. Right click to create the freehand sketch.

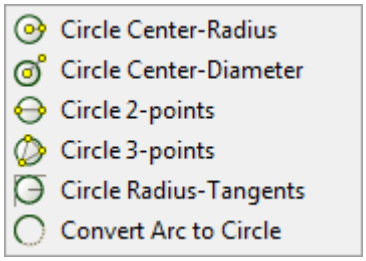
Command Options

Keyboard	Prompt Box	Description
D	Delete on/off	Toggle between the Pencil tool () and the Erase tool (). When the Erase tool is active, move the cursor over the freehand sketch line to erase segments. Because the Erase tool cannot break a freehand sketch line, always begin at the start or end segment of the freehand sketch line to erase.
C	Connect to end	After erasing segments, move the cursor to the last segment to continue.
W	Write, then resume	Create the freehand sketch line(s) drawn so far, then continue the Freehand command.
P		Toggles between *pen up* (move cursor without drawing) and *pen down* (draw segments).
S		Straight to cursor: connect the endpoint of the last segment to the current cursor position when pen is up.
Enter	Done	Create the freehand sketch line(s) and conclude the Freehand tool.

Circles

Circles

The default method for drawing a circle is to specify a center point and radius.
Other methods to draw circles can be found in the *Circle* submenu of the *Draw* menu.

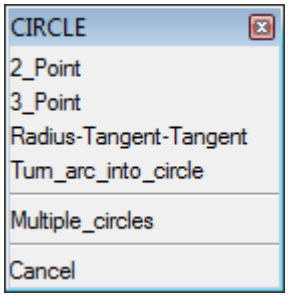


General procedure to draw a circle

1. Do one of the following.

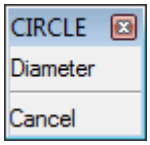
- Click the *Circle* tool button ( ) on the *Draw 2D* toolbar.
- Type *circle* in the command bar, then press Enter.
- Type *C* in the command bar, then press Enter.

The command bar reads: 2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>:
A prompt box displays:



2. Specify the center point of the circle.

The command bar reads: Diameter/<Radius> <current radius>:
The prompt box changes:



3. Do one of the following:

- Press Enter to accept the current radius.
- Type a new radius and press Enter.
- Click to define the radius graphically.

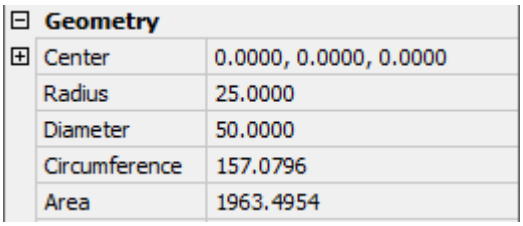
Command Options

Keyboard	Prompt Box	Description
2P	2_Point	Draw a circle by defining the diameter by two points.
3P	3-Point	Draw a circle by specifying three points.
RTT	Radius-Tangent-Tangent	Draw a circle tangent to two entities and a specified radius. You can draw circles tangent to lines, polyline segments, arcs and circles.
A	Turn_arc_into_circle	Turn an arc into a circle.
M	Multiple_Circles	Create multiple circles of the same size.
D	Diameter	Draw a circle by specifying the center point and diameter.
Esc	Cancel	Abort the Circle command.

Editing a circle

1. Select the circle.

   The properties of the circle display in the Bricscad Properties bar.



2. Type a new value in the *Center*, *Radius*, *Diameter*, *Circumference* or *Area* field, then press Enter.

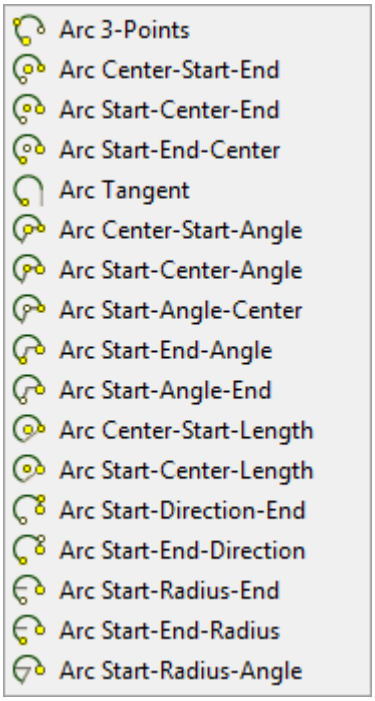
3. Press the Esc key to stop editing the circle.

Arcs

Arcs

The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint.

Other methods to draw arcs can be found in the *Arc* submenu of the *Draw* menu.



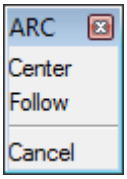
General procedure to draw an arc

1. Do one of the following.

- Click the *Arc* tool button ( ) on the *Draw 2D* toolbar.
- Type *arc* in the command bar, then press Enter.
- Type *A* in the command bar, then press Enter.

The command bar reads: Enter to use last point/Center/Follow/<Start of arc>:

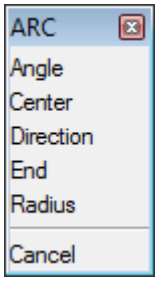
A prompt box displays:



2. Specify the start point of the arc.

The command bar reads: Angle/Center/Direction/End/Radius/<Second point>:

The prompt box changes:

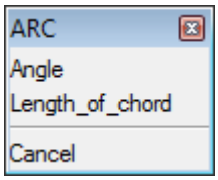
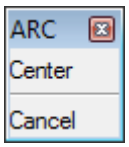


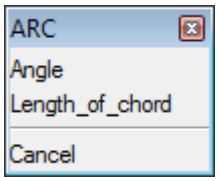
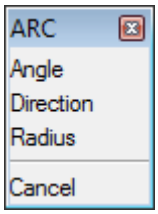
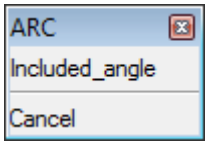
3. Specify the second point of the arc.

The command bar reads: End point:

4. Specify the end point of the arc.

Command Options

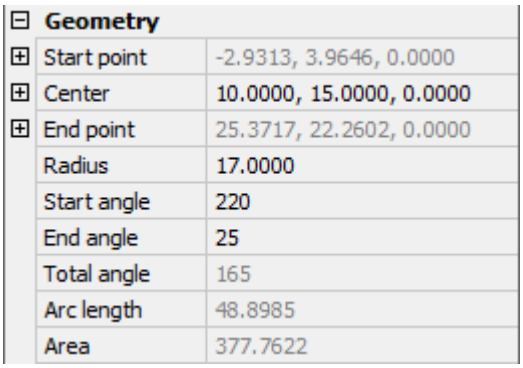
Keyboard	Prompt Box	Description
C	Center	Draw an arc by first specifying the center point, then the start point, then you can choose between specifying: • endpoint (default) • the included angle (option) • the length of the chord (option) 
F	Follow	Draw an arc tangent to the previously drawn line or polyline segment. The arc start at the endpoint of the line or polyline segment.
Esc	Cancel	Abort the Arc tool.
The following options are available after specifying the start point of the arc first:		
A	Angle	Specify the included angle, then you can choose between specifying: • the endpoint (default) • the center point (option) 
C	Center	Specify the center point, then you can choose between specifying: • the endpoint (default) • the included angle (option) • the length of the chord (option)

		
D	Direction	Define the direction of tangent in the start point of the arc, then define the endpoint.
E	End	Define the end point of the arc, then you can choose between specifying: • center point (default) • the included angle (option) • the direction of the tangent in the start point of the arc (option) • the radius (option) 
R	Radius	Define the radius, then you can choose between specifying: • the endpoint (default) • the included angle (option) 
Esc	Cancel	Abort the Arc tool.

Editing an arc

1. Select the arc.

   The properties of the arc display in the Bricscad Properties bar.



Geometry	
Start point	-2.9313, 3.9646, 0.0000
Center	10.0000, 15.0000, 0.0000
End point	25.3717, 22.2602, 0.0000
Radius	17.0000
Start angle	220
End angle	25
Total angle	165
Arc length	48.8985
Area	377.7622

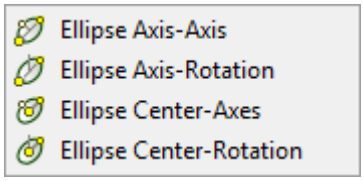
2. Type a new value in the *Center*, *Radius*, *Start angle* or *End angle*, then press Enter.
3. Press the Esc key to stop editing the arc.

Ellipses

Ellipses

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

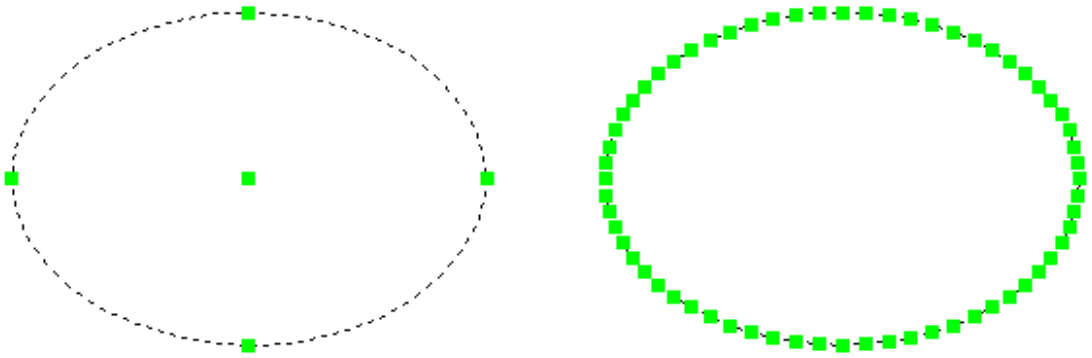
Other methods to draw ellipses can be found in the *Ellipse* submenu of the *Draw* menu.



The *Polyline Ellipse* variable: controls whether real ellipses or a polyline representation is created.

If the variable is ON, the Ellipse tool creates a closed polyline, if the variable is OFF real ellipses are created.

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



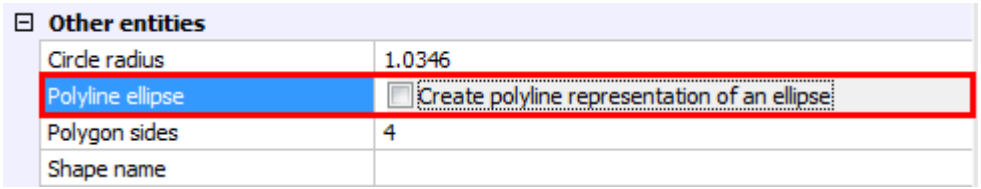
real ellipse (left) and the polyline representation of an ellipse (right)

NOTE Since ellipses cannot be used as a clip boundary for external references or to create clipped paperspace viewports, you must use a polyline representation to do these jobs.

Setting the Polyline Ellipse variable

Do one of the following:

- In the command bar type *pellipse*, then press Enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Other entities*.

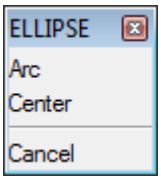


General procedure to draw an ellipse

1. Do one of the following.

- Click the *Ellipse* tool button ( ) on the *Draw 2D* toolbar.
- Type *ellipse* in the command bar, then press Enter.
- Type *EL* in the command bar, then press Enter.

The command bar reads: Arc/Center/<First end of ellipse axis>:
A prompt box displays:

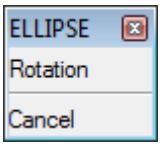


2. Specify the the first end of the ellipse axis.

The command bar reads: Second end of axis:

3. Specify the second end of the ellipse axis.

The command bar reads: Rotations/<Other axis>:
A prompt box displays:



4. Specify the length of half the other axis of the ellipse.

Command Options

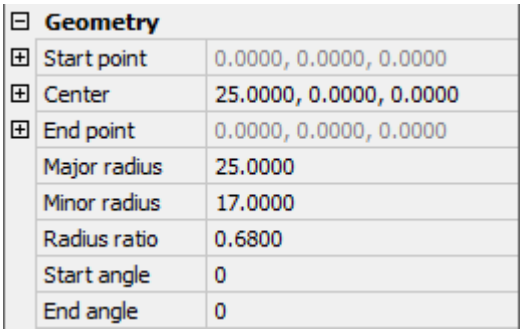
Keyboard	Prompt Box	Description
A	Arc	Creates an elliptical arc.
C	Center	First specify the center point of the ellipse, then specify the endpoint of the first axis and the length of half the second axis.
R	Rotation	Specify the rotation around the first axis.
Esc	Cancel	Abort the Ellipse command.

NOTE If the *Polyline Ellipse* variable is off, the *Arc* option is not available.

Editing an ellipse

1. Select the ellipse.

   The properties of the ellipse display in the Bricscad Properties bar.



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

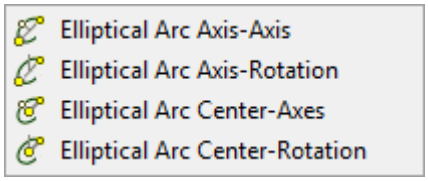
3. Press the Esc key to stop editing the ellipse.

Elliptical Arcs

Elliptical Arcs

An elliptical arc is a portion of an ellipse. The default method for drawing an elliptical arc is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis.

Other methods to draw elliptical arcs can be found in the *Ellipse* submenu of the *Draw* menu.



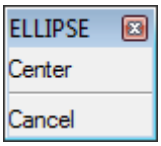
General procedure to draw an elliptical arc

1. Do one of the following.

- Click the *Elliptical Arc* tool button ( ) on the *Draw 2D* toolbar.
- Launch the Ellipse command using the *Arc* option.

The command bar reads: Center/<First end of ellipse axis>:

A prompt box displays:



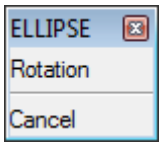
2. Specify the first end or the ellipse axis.

The command bar reads: Second end of axis:

3. Specify the second end of the ellipse axis.

The command bar reads: Rotation/<Other axis>

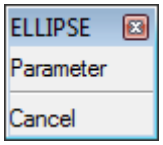
A prompt box displays:



4. Specify the length of half the other axis of the ellipse.

The command bar reads: Parameter/<Start angle of arc>:

The prompt box changes:

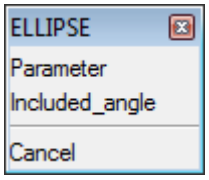


5. Specify the start angle of the elliptical arc.

The elliptical arc is drawn dynamically when you move the crosshairs.

The command bar reads: Parameter/Included/<End angle>:

The prompt box changes:



6. Specify the end angle of the elliptical arc.

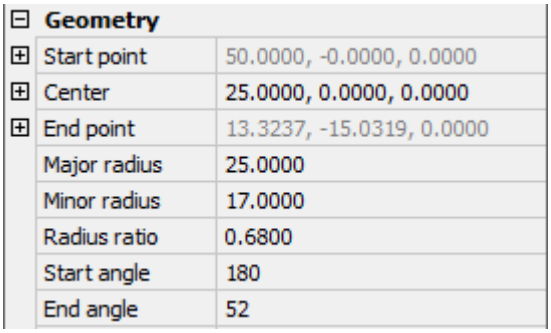
Command Options

Keyboard	Prompt Box	Description
C	Center	First specify the center point of the ellipse, then specify the endpoint of the first axis and the length of half the second axis.
R	Rotation	Specify the rotation around the first axis.
P	Parameter	Draw an elliptical arc by specifying its parametric vector equation. The equation is:p(u) = c + a * cos(u) = b * sin(u) c = the center of the ellipse; a = major axis; b = minor axis
I	Included angle	Angle between the radius vector of the start point and the radius vector of the end point of the elliptical arc.
Esc	Cancel	Abort the Elliptical Arc tool.

Editing an elliptical arc

1. Select the elliptical arc.

   The properties of the elliptical arc display in the Bricscad Properties bar.



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

3. Press the Esc key to stop editing the elliptical arc.

Rectangles

A rectangle is created as a closed, four sided polyline. A rectangle is drawn by specifying two opposite corners. Unless the Rotated option is chosen, the sides of a rectangle are always parallel to the x-axis and y-axis of the current coordinate system.

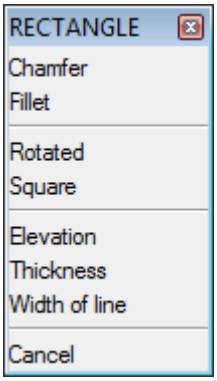
Drawing rectangles

1. Do one of the following

- Click the Rectangle tool button ( ) on the *Draw 2D* toolbar.
- Choose Rectangle in the *Draw* menu.
- Type *rectangle* in the command bar, then press Enter.
- Type *rec* in the command bar, then press Enter.

The command bar reads:
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<select first corner of rectangle>:

A prompt box opens:



2. Specify the first corner of the rectangle.

The rectangle displays dynamically when you move the cursor.

3. Specify the opposite corner of the rectangle.

The rectangle is created.

4. (option) Right click to draw more rectangles.

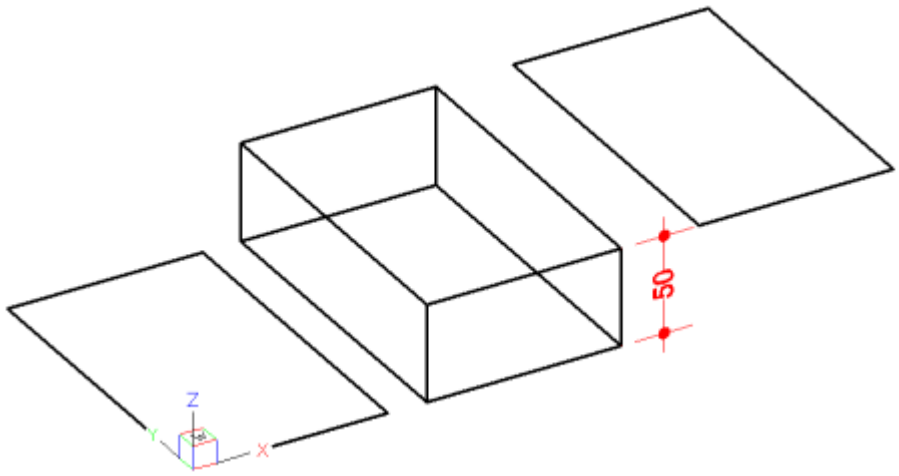
NOTE Because a rectangle is created as a closed polyline, rectangles have a direction. The direction of a closed polyline is positive if it is drawn counter-clockwise and negative if it is drawn in a clockwise direction. Rectangles have a counter-clockwise direction if both the X- and Y-coordinate of the second point are higher or lower than the coordinates of the first point (= movement of the cursor is positive or negative along X- and Y-axis). Rectangles have a clock-wise direction if the X- or Y-coordinate is higher or lower than the X- or Y-coordinate of the first point (= movement of the cursor is positive along the X-axis and negative along the Y-axis or vice versa).

The *Reverse Direction* option of the Edit Polyline tool changes the direction of a polyline.

Command Options

Keyboard	Prompt	Description

	Box	
C	Chamfer	Draw a rectangle of which the corners are equally chamfered. You are prompted to specify the chamfer distances.
F	Fillet	Draw a rectangle of which the corners are equally filleted. You are prompted to specify the fillet radius.
R	Rotated	Draw a rotated rectangle. When the second corner is specified, you are prompted to type the rotation angle in the command bar or define the angle graphically.
S	Square	Draw a square. Use Ortho Mode to draw the square parallel to the x-axis. Squares can also be drawn as a four-sided polygon.
E	Elevation	Draw a rectangle at the specified *Elevation*. All rectangles created in the current session of the drawing will be drawn with an offset from the xy-plane, until you define the *Elevation* differently.
T	Thickness	Define the *Thickness* for the rectangle polyline
W	Width of line	Define the *Width* for the four segments of the rectangle polyline.
Esc	Cancel	Abort the Rectangle command.



Rectangles: Elevation = 0 (left), Thickness = 50 (middle), Elevation = 50 (right)

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Polygons

Polygons are closed polylines comprised of a minimum of three and a maximum of 1,024 equal-length sides.

In Bricscad you can draw polygons using the following methods:

- *Centre - Vertex*: first define the center point, then the vertex (= radius of the circumscribed circle method).
- *Center - Side*: first define the center point, then the midpoint of a side (= radius of the inscribed circle method).
- *Edge*: define the length of the side of the polygon.

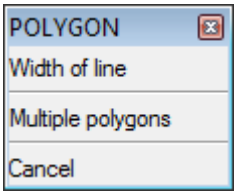
Drawing polygons

1. Do one of the following

- Click the Polygon, Center - Vertex tool button ( ) on the *Rectangle* flyout of *Draw 2D* toolbar.
- Choose Polygon in the *Draw* menu.
- Type *polygon* in the command bar, then press Enter.

The command bar reads: Polygon: Multiple/Width of line/<Number of sides> <4>:

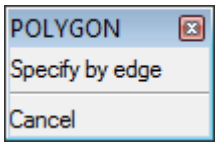
A prompt box opens:



2. Specify the number of sides, then press Enter.

The command bar reads: Specify by: Edge/<Center of polygon>:

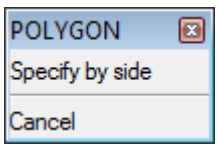
The prompt box changes:



3. Specify the center of the polygon.

The command bar reads: Specify by: Side/<Select vertex point>:

the prompt box changes:



4. Specify a vertex point of the polygon.

The polygon is created.

Command Options

Keyboard	Prompt Box	Description
W	Width of line	Define the *Width* for the all segments of the polygon.
M	Multiple polygons	Draw multiple identical polygons. When the polygon is created you are prompted to define the midpoints of the other polygons. This option is not available using the *Edge* method.
E	Specify by edge	Use the *Edge* method.
V	Specify by vertex	Use the *Center - Vertex* method.
S	Specify by side	Use the *Center - Side* method.
Esc	Cancel	Abort the Polygon command.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Traces

Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.

Drawing traces

1. Do one of the following

- Click the Trace tool button () on the *Rectangle* flyout of the *Draw 2D* toolbar.
- Choose Trace in the *Draw* menu.
- Type *trace* in the command bar, then press Enter.

The command bar reads: Width of trace <current width>:

2. Do one of the following:

- Right click to accept the current width.
- Type a new current width in the command bar.
- Define a new current width graphically by clicking two points.

The command bar reads: Start of trace:

3. Specify the start point of the trace.

The command bar reads: Next point:

4. Specify the end point of the first trace.

A line indicates the direction of the first trace. The direction of the second trace displays dynamically.

5. Do one of the following:

- Right click to create the trace.
- Specify the endpoint of the next trace.

6. (option) Repeat step 5 to keep adding traces.

NOTE When Fill Mode is turned off, all filled entities, such as traces, display and print as outlines.

Donuts

Donuts are solid, filled circles or rings created as closed, 2-segment, wide polylines. The default method to draw donuts is to specify its inside and outside diameters, and then specify its center. If the inside diameter is zero, a filled circle is created.

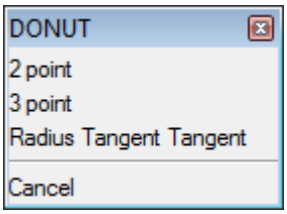
Drawing donuts

1. Do one of the following:

- Click the Donut tool button ( ) on the *Rectangle* flyout of *Draw 2D* toolbar.
- Choose Donut in the *Draw* menu.
- Type *donut* in the command bar, then press Enter.

The command bar reads: 2Point/3Point/RadTanTan/<Inside diameter of donut> <current inside diameter>:

A prompt box opens:



2. Do one of the following:

- Right click to accept the current inside diameter.
- Type a new inside diameter in the command bar.
- Define a new inside diameter graphically by clicking two points.

The command bar reads: Outside diameter of donut <current outside diameter>:

3. Do one of the following:

- Right click to accept the current outside diameter.
- Type a new outside diameter in the command bar.
- Define a new outside diameter graphically by clicking two points.

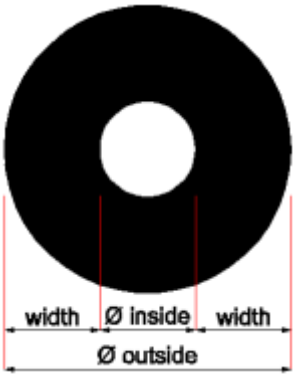
The command bar reads: Center of donut:

4. Specify the center of the donut.

The donut is created.

5. (option) Specify the center point of another donut.

6. Right click to conclude the donut command.



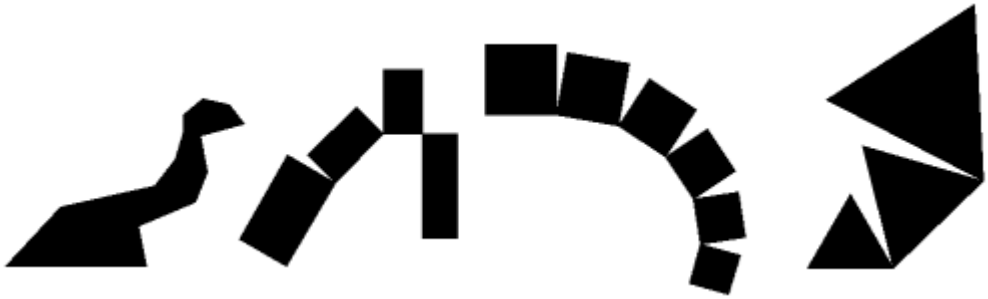
Command Options

Keyboard	Prompt Box	Description
2P	2 point	Draw a donut by specifying the width and outside diameter
3P	3 point	Draw a donut by first specifying the width, then you are prompted to specify three points on the outside of the donut.
RTT	Radius Tangent Tangent	Draw a donut tangent to 2 circles or circular arcs. You are prompted to specify the width and outside diameter of the donut.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Planes

Planes are rectangular, square, triangular or quadrilateral areas filled with a solid color. Planes in Bricscad are similar to Solids in Autocad®.



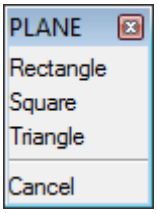
Left to right: quadrilateral, rectangular, square and triangular planes

Drawing planes

1. Do one of the following:

- Click the Plane tool button ( ) on the *Rectangle* flyout of *Draw 2D* toolbar.
- Choose Plane in the *Draw* menu.
- Type *plane* in the command bar, then press Enter.

  The command bar reads: Rectangle/Square/Triangle/<First point of plane>:
  A prompt box opens:



2. Specify the first point of the plane.
3. Specify the second point of the plane.
4. Specify the third and fourth point of the plane.

The plane is created.

5. (option) Repeat step 4 to add more planes.

A series of connected planes is created.

6. Right click to conclude.

Command Options

Keyboard	Prompt Box	Description
R	Rectangle	Draw a rectangular plane. You are prompted to specify two opposite corners of the plane and a rotation angle (much like in the *Rotated* option of the Rectangle command). You can create a series of connected rectangular planes.

S	Square	Draw a square plane. You are prompted to specify two subsequent corners of the square (much like the *Square* option of the Rectangle command). You can create a series of connected square planes.
T	Triangle	Draw a equal sided triangular plane. You are prompted to specify to corners of the triangle. You can create a series of connected triangular planes.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Revision Clouds

A revision cloud is an open or closed polyline composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated.

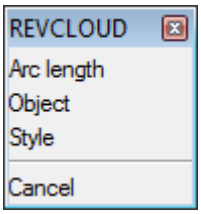
Before you start drawing revision clouds, first define the *Arc length* and *Style*.

Drawing revision clouds

1. Do one of the following:

- Click the Revision Cloud tool button ( ) on the *Rectangle* flyout of *Draw 2D* toolbar.
- Choose Revision Cloud in the *Draw* menu.
- Type *revision cloud* in the command bar, then press Enter.

The command bar reads: Specify start point or [Arc length/Object/Style] <Object>:
A prompt box opens:



2. (option) To define the *Arc length*, do one of the following:

- Choose *Arc length* in the prompt box.
- Type *A* in the command bar, then press Enter.

The command bar reads: Specify minimum length of arc <current minimum length>:
Do one of the following:

- Press Enter or right click to accept the current minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new minimum length graphically by specifying two points.

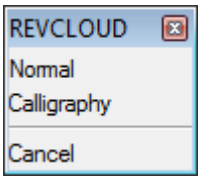
The command bar reads: Specify maximum length of arc <current minimum length>:
Do one of the following:

- Press Enter or right click to maximum length equal to the minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new maximum length graphically by specifying two points.

3. (option) To define the revision cloud *Style*, do one of the following:

- Choose *Style* in the prompt box.
- Type *S* in the command bar, then press Enter.

The command bar reads: Select arc style [Normal/Calligraphy] <current style>:
A prompt box opens:



Do one of the following:

- Press Enter or right click to accept the current style
- Select a new style in the prompt box.
- Type *N* or *C* in the command bar, then press Enter.

4. (option) To convert an existing polyline, arc or circle to a revision cloud, do one of the following:

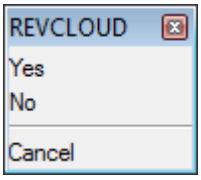
- Press Enter, then select the entity you want to convert.
- Choose Object in the prompt box, then select the entity you want to convert.

5. (option) To draw the revision cloud, click where you want to start and move the cursor. Each time the cursor movement exceeds the minimum length, an arc is added.

To stop do one of the following:

- Move the cursor over the start point to close the revision cloud.
- Right click.

The command bar reads: Reverse direction [Yes/No] <No>:
A prompt box opens:



6. To create the revision cloud, do one of the following:

- To accept the revision cloud, right click.
- To reverse the direction, type *Y* or choose *Yes* in the prompt box.
- To leave the command without creating the revision cloud, choose *Cancel* in the prompt box or press the Esc key.

Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle)

Points

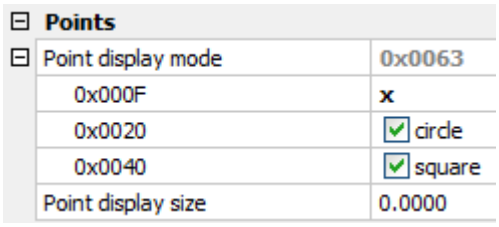
Points

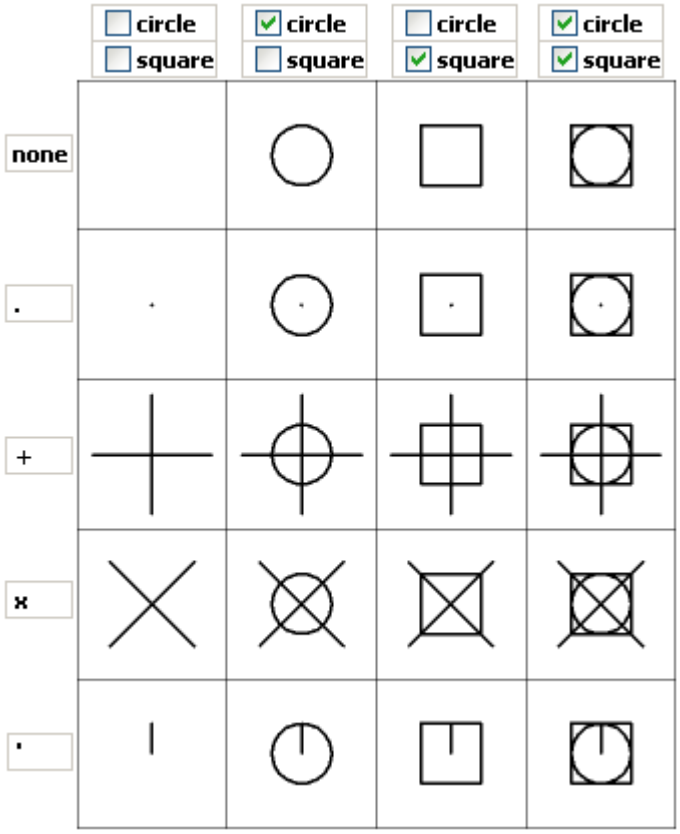
Since points are dimensionless, a single pixel should be the correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles.

The point display style is chosen by means of the *Point Display Mode* (PDMODE) setting. The size of the point representation style is controlled through the *Point Display Size* (PDSIZE) setting.

Defining the point display settings

1. Open the Settings dialog.
2. In the *Drawing* settings category, expand the *Entity Creation* sub-category.
3. Expand the *Points* settings group, then expand the *Point display mode* setting.





Point display modes

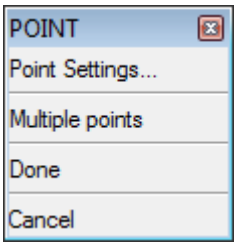
4. Set the *Point Display Mode* setting:
   - Choose the *point location marker* style: *point*, *none*, *X* or |
   - Choose to add a *circle* and/or a *square*.
5. Set the *Point Display Size* setting.
   - Relative: Type 0 (zero) to scale the point display style at 5% of the screen.
   - Absolute: Type the size of the point display style in drawing units.
6. Close the *Settings* dialog.

NOTE If you need to draw a 'dot', use a Donut with a zero inside diameter instead of a point entity.

Drawing points

1. Do one of the following
   - Click the Point tool button ( ) on the *Draw 2D* toolbar.
   - Choose Point in the *Draw* menu.
   - Type *point* in the command bar, then press Enter.

   The command bar reads: Settings/Multiple/<Location of point>:
   A prompt box opens:



2. Specify the location of the point.

The point is created.

Command Options

Keyboard	Prompt Box	Description
S	Point Settings	Define the point settings.
M	Multiple Points	Draw multiple points.
D	Done	When drawing multiple points, stop drawing points.
C	Cancel	Abort the point command.

NOTE If the *Point Display Size* setting is set to 0 (zero), use the Regen tool to resize the display of the points to 5% of the screen size after zooming in or out.

Working with hatches

Hatching Overview

When you add hatching to a drawing, Bricscad fills entities or enclosed areas with a predefined pattern or lines. First you specify the hatch pattern and other options, and then you choose which entities or enclosed areas that you want to hatch.

NOTES

- Hatch patterns are memory intensive and can take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing.
- Hatches that are too dense, are not displayed. If such hatches exist in a drawing, a warning displays in the command bar after opening the drawing. The maximum number of dashes is controlled by the MAXHATCH setting. By default the maximum number of dashes is 100 000.
- The hatch command is obsolete and still exists for compatibility only. Use Boundary Hatch (bhatch) instead. Boundary Hatch creates hatching entities, which can be edited afterwards, while hatch creates anonymous, static blocks.

The pattern files sit in the Support folder of the current user.

The *MEASUREMENT* setting controls which hatch pattern file will be used.

- *Imperial*: uses Default.pat
- *Metric*: uses Iso.pat

NOTE Custom hatch pattern files are not yet supported, though custom hatch patterns can be added in the Default.pat and Iso.pat files.

Defining the MEASUREMENT setting

Do one of the following:

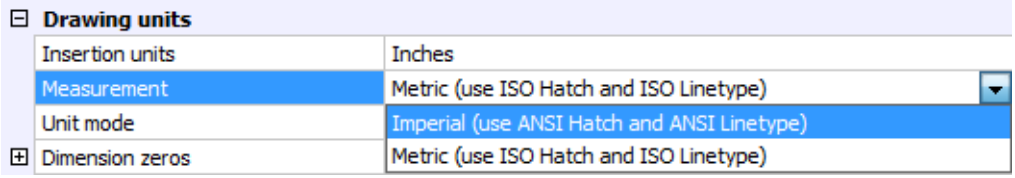
- In the command bar type *measurement*, then press Enter.

  Type *ON*, then press Enter to set *measurement* to *Metric*.

  Type *OFF*, then press Enter to set *measurement* to *Imperial*.

- In the Settings dialog go to *Drawing > Drafting > Drawing units.*

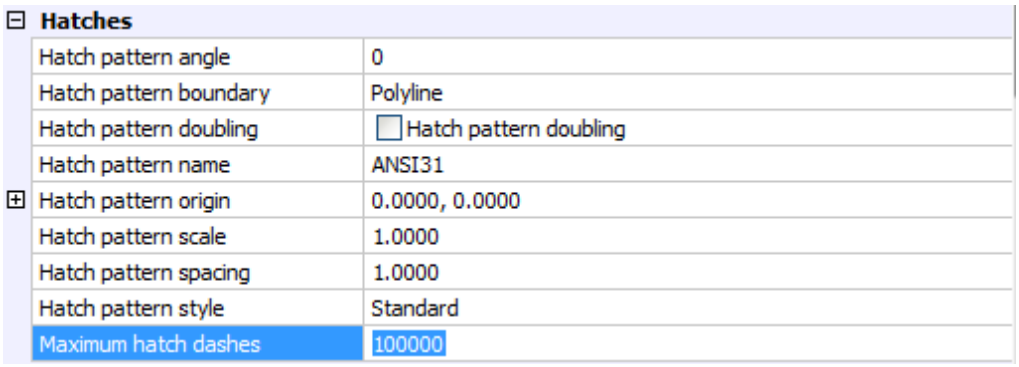
  Choose either *Metric* or *Imperial* in the combo box.



Defining the MAXHATCH setting

Do one of the following:

- In the command bar type *maxhatch*, then press Enter.
  Type a new value and press enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Hatches*.
  Type a new value in the Maximum hatch dashes settings field.



Using Boundary Hatch

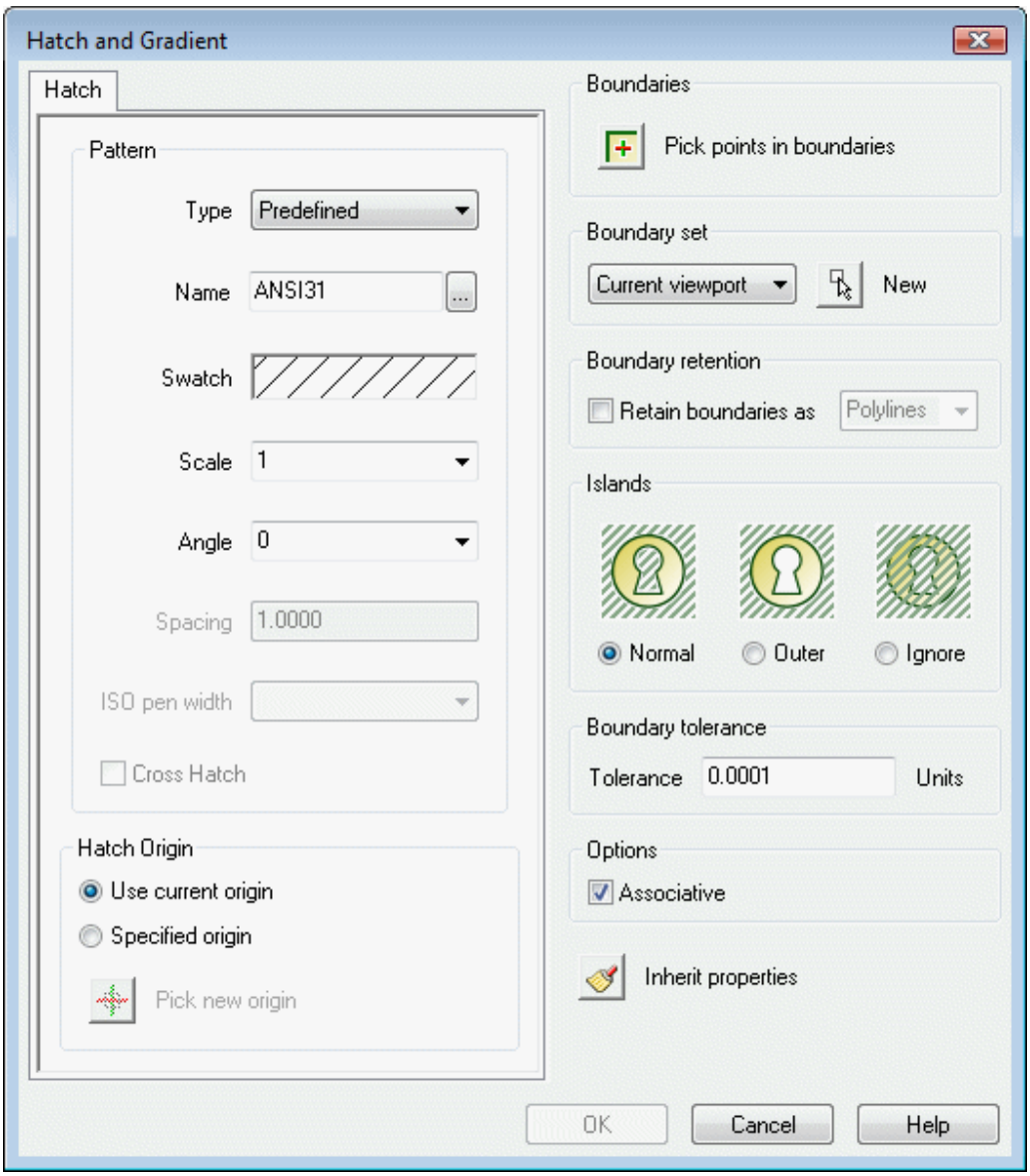
The Boundary Hatch tool fills an enclosed area in your drawing with a pattern. The area can either be a single entity such as a circle or a closed polyline or a selection of entities. To create the hatch you must click inside the closed perimeter of a boundary, not on one of the boundary entities. If no entities are selected, Bricscad will automatically detect the boundary entities.

General procedure to create hatching

1. Do one of the following:

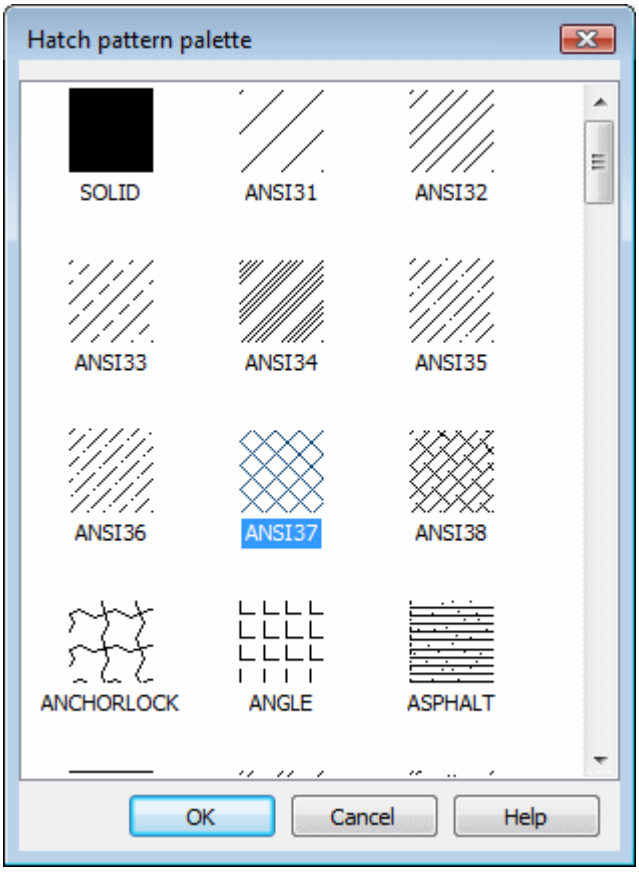
- Click the Boundary Hatch tool button ( ) on the *Draw 2D* toolbar.
- Choose Boundary Hatch... in the *Draw* menu.
- Type *bhatch* in the command bar, then press enter.

The *Hatch and Gradient* dialog opens:



2. (option) Select the *Pattern Type*.

3. (option) If the *Pattern Type* is *Predefined*, select a *Pattern Name* in the *Hatch pattern* palette,
then double click the pattern or click the *OK* Button.



4. (option) If the *Pattern Type* is *Predefined*, do one of the following:

- Type a *Scale* in the *Scale* settings field.
- Choose the *Scale* from the drop down list.

5. (option) Do one of the following:

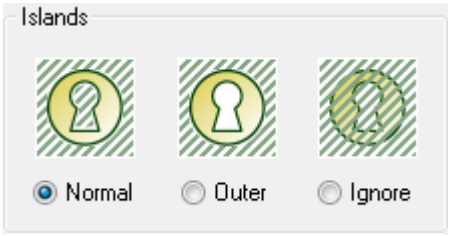
- Type a *Angle* in the *Angle* settings field.
- Choose the *Angle* from the drop down list.

6. (option) If the *Pattern Type* is *User defined*, specify the *Pattern Spacing*.

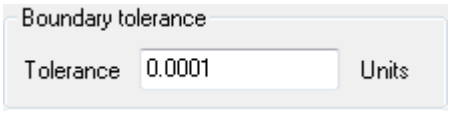
7. (option) If the *Pattern Type* is *User defined*, select the *Cross Hatch* option.

8. (option) Select the *Boundary retention* option.

9. (option) Select an *Island* option.



10. (option) Adjust the *Boundary tolerance*.



11. (option) Select the *Associative* option.

12. (option) Click the *New* button () to select a *Boundary set*.

The *Hatch and Gradient* dialog temporarily closes to let you select entities.

(see also the note below)

13. (option) Click the *Specified Origin* radio button, then click the *Pick a new origin* button ().

The *Hatch and Gradient* dialog temporarily closes to let you specify a new origin.

14. Click the Pick Points button ().

The *Hatch and Gradient* dialog closes.

The command bar reads: Select a point to define a boundary or hatch area:

15. Click in the area you want to hatch.

The command bar reads: Select a point to define a boundary or hatch area:

16. (option) Repeat step 14 to hatch more areas.

17. Right click or press Enter to stop adding areas.

The *Hatch and Gradient* dialog displays again.

18. (option) Click the *Specified Origin* radio button, then click the *Pick a new origin* button ().

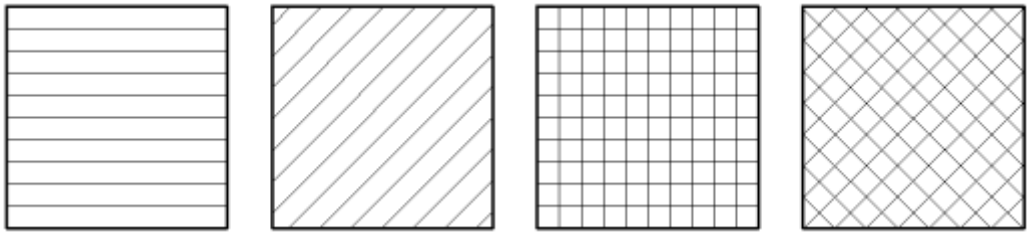
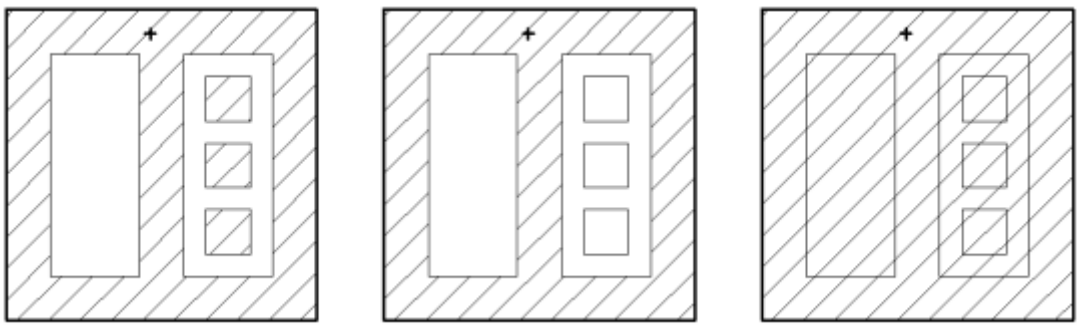
The *Hatch and Gradient* dialog temporarily closes to let you specify a new origin.

19. Click the *OK* button to create the hatching.

NOTE If a selection set was active when you launch the *Boundary Hatch* tool, this selection will be used as the *Boundary Set* if you click the *New* button in step 11 in the above procedure. In this case the *Hatch and Gradient* dialog closes and immediately reopens.

Command Options

Option	Description
Pattern Type	User defined: the pattern type consists of parallel lines at a fixed distance. Predefined: the hatching consists of a predefined repeated pattern, read from a pattern file.
Pattern Name	The name of a predefined hatch pattern, as defined in the pattern file.

Pattern Scale	Scaling factor used for predefined patterns.
Pattern Angle	Rotation angle of the hatching. The pattern angle setting applies to both user defined and predefined patterns.
Pattern Spacing	Distance between the parallel lines of a user defined pattern.
Cross Hatch	Repeats a user defined pattern with the same spacing and origin, but rotated 90° with respect to the pattern angle.  single hatching (left) cross hatching (right)
Hatch origin	Start point of the first line of a user defined hatch pattern. Origin of the first instance of a predefined hatch pattern. The hatch origin is relative to the current coordinate system (UCS or WCS)
Boundary Set	Selection of entities to take part in the calculation of a hatch boundary. If no *Boundary set* is active, all entities that are visible in the current viewport will be taken into account. In a complex drawing it is recommended to use a *Boundary set* in order to limit the calculation time.
Boundary Retention	If the boundary of the hatch is composed of multiple entities, you can use this option to create a new boundary entity. If the hatch boundary is a single entity already, a new entity is created on top of the original entity.
Islands	If within the detected enclosed area other closed areas exist (so called islands), you can choose between three options: • *Normal*: creates hatching inside all closed areas found. • *Outer*: creates hatching between the outer boundary and the first islands only. • *Ignore*: ignore all islands  When you click at the location of the cross, the result of the island options are: normal (left) outer (middle) ignore (right).
Boundary Tolerance	The Boundary Hatch tool searches for an enclosed area around the point that you specify in the drawing. The Boundary Tolerance setting defines the maximum size of gaps in the boundary.
Associative	An associative hatching is connected to the boundary entities. If you move one of the boundary entities, the hatching is recalculated. You can remove the associativity afterwards, but you cannot turn a non-associative hatch into a associative one.

Inherit properties	Reads the properties of an existing hatch, in order to reuse them to create a new hatching.

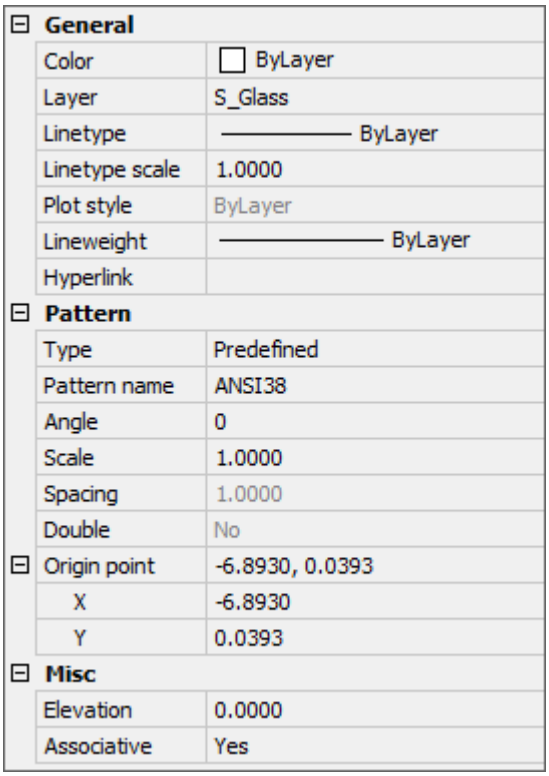
Editing a hatch

If you select a hatch, its properties display in the Bricscad Properties bar.

To edit a hatch

1. Select the hatch in the drawing.

   The properties of the selected hatch display.



General	
Color	ByLayer
Layer	S_Glass
Linetype	ByLayer
Linetype scale	1.0000
Plot style	ByLayer
Lineweight	ByLayer
Hyperlink	
Pattern	
Type	Predefined
Pattern name	ANSI38
Angle	0
Scale	1.0000
Spacing	1.0000
Double	No
Origin point	-6.8930, 0.0393
X	-6.8930
Y	0.0393
Misc	
Elevation	0.0000
Associative	Yes

2. Select the property you want to modify in the Bricscad Properties bar.
3. Modify the selected property
4. Do one of the following:

   - Repeat steps 2 and 3 to modify another property.
   - Press the Esc key to stop editing the hatch.

NOTES

- If you select multiple hatches you can you edit them simultaneously. Properties which are different display as **varies** in the Bricscad Properties bar. If you edit such property, all edited hatches will then share this property.
- Linetype, Linetype scale and Lineweight only apply to *User Defined* pattern types.

Modifying Entities

Overview

Modifying Commands Overview

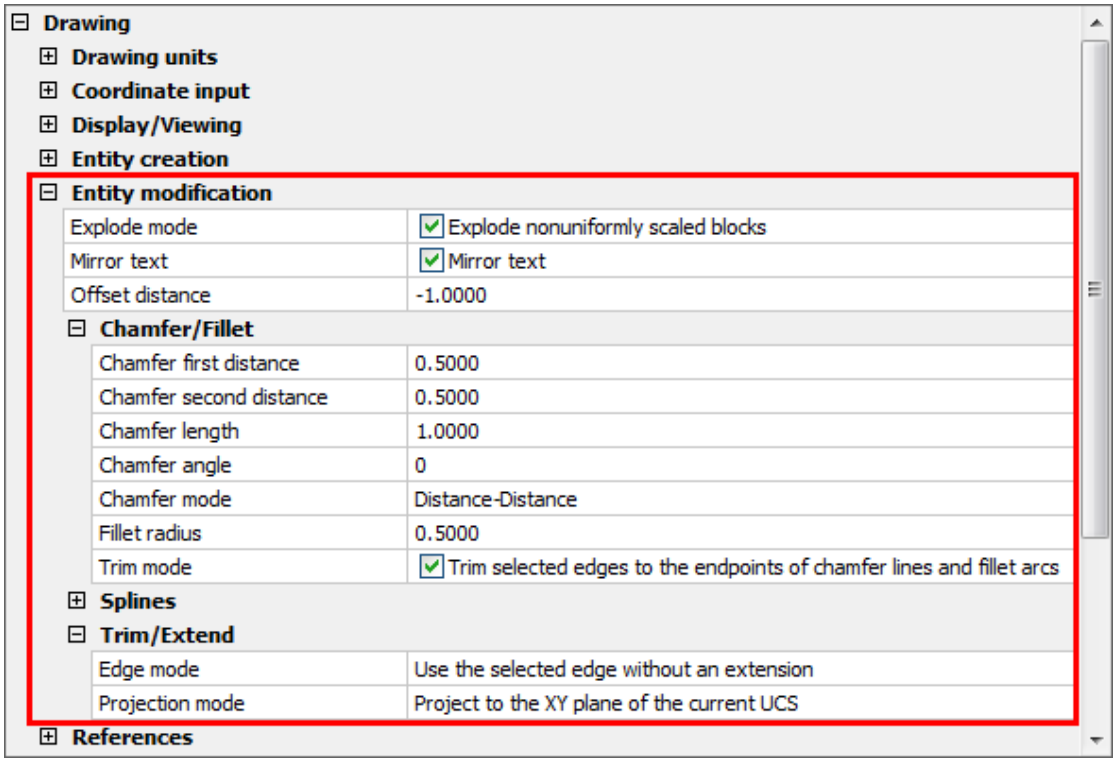
All entity modification tools can be found either on the *Modify* toolbar or in the *Modify* menu.

Icon	Tool name	Keyboard	Description
	Move	M	Moves entities to another location in the same drawing or into another drawing.
	Copy	CO	Draws a duplicate or multiple duplicates of the selected entities.
	Parallel	PARALLEL	Creates a parallel or offset copy of curves and lines.
	Scale	SC	Changes the scale of existing entities, either enlarging them or reducing them proportionately in x, y, and z directions.
	Rotate	RO	Rotates entities around a specified point.
	3D Rotate	ROTATE3D	Rotates entities around a three-dimensional axis.
	Mirror	MI	Moves or copies the reflected image of entities about a line.
	3D Mirror	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	Array	AR	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	3D Array	3DARRAY	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	Break	BR	Splits an entity into two entities.
	Join	JOIN	Joins two entities (two or more lines or two or more arcs) into one entity.
	Trim	TR	Erases the portions of selected entities that cross a specified boundary.
	Flatten	FLATTEN	Equals the Z-coordinate of the selected entities.
	Align	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	Extend	EX	Lengthens lines, arcs, polylines or rays to meet another entity.

	Edit Length	EDITLEN	Changes the length of a line, polyline, freehand entity or arc.
	Stretch	S	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	Measure	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	Divide	DIV	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.
	Chamfer	CHA	Creates a chamfer, or a beveled edge, at the intersection of two 3D solids, lines, rays, or infinite lines.
	Fillet	F	Creates a fillet, or rounded corner, at the intersection of 3D solids, two lines, rays, or infinite lines.
	Edit Polyline	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	Edit Text	DDEDIT	Modifies the properties of selected text entities. See Working with Text.
	Explode	EXPLODE	Ungroups a block, polyline, polyface mesh, solid, or hatch, creating separate entities for each element.
	Region	REG	Converts a closed entity into a two-dimensional region.
	Change	CHANGE	Changes the location, size, color, elevation, layer, linetype, linetype scale, lineweight, and three-dimensional thickness of entities.
	Properties	PROPERTIES	Opens the Bricscad Properties Bar (if not already open).

Entity Modification Settings

Entity modification settings are found in the *Entity modification* settings group of the *Drawing* settings category in the Settings Manager.

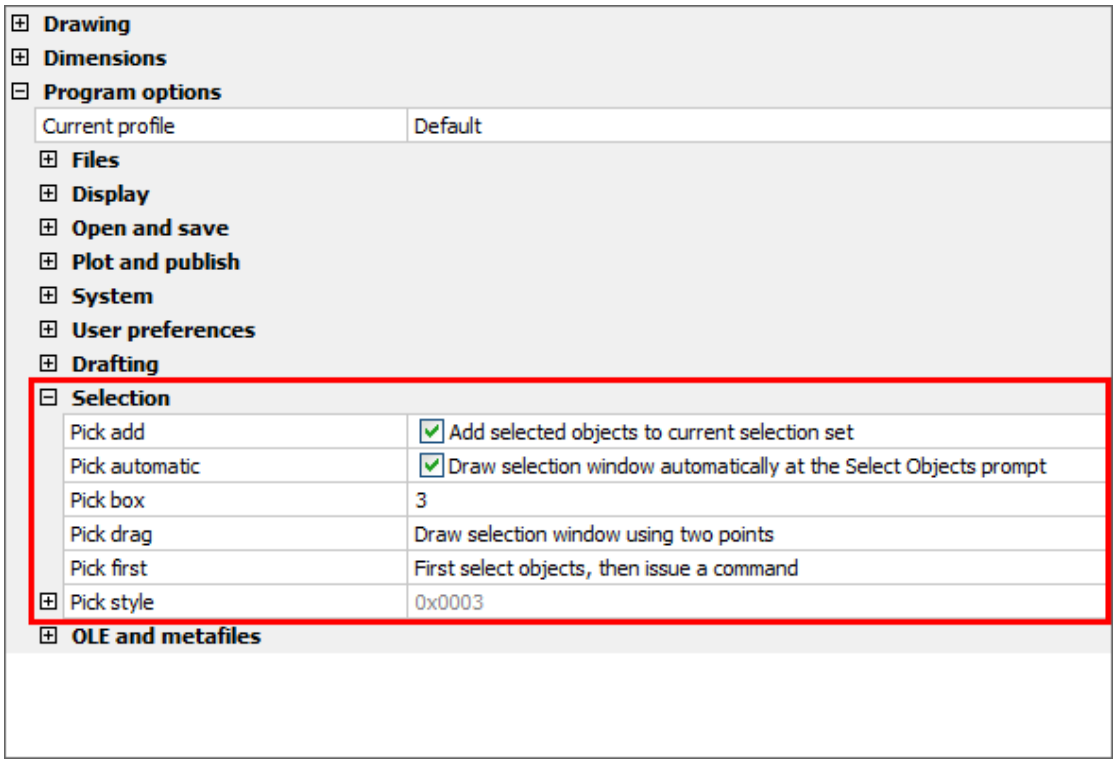


Name	Command	Description
Explode Mode	EXPLMODE	Controls whether explode supports non-uniformly scaled blocks
Offset Distance	OFFSETDIST	Sets the default offset distance
Offset Gap Type	OFFSETGAPTYPE	Controls how to offset polylines when a gap is created as a result of offsetting the individual polyline segments.
Chamfer Mode	CHAMMODE	Sets the input method by which chamfers are created.
Trim Mode	TRIMMODE	Controls whether selected edges for chamfers and fillets are trimmed.
Edge Mode	EDGEMODE	Controls how TRIM and EXTEND determine cutting and boundary edges.
Projection Mode	PROJMODE	Sets the current projection mode for TRIM and EXTEND.

Selecting Entities

Selection Settings

Selection settings are found in the *Selection* settings group of the *Program Options* settings category in the Settings Manager.



Name	Command	Description
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. *ON*: The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. *OFF*: You cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the *Shift* key, you can add entities. If you select an entity that was already selected while pressing the *Shift* key, this entity is removed from the selection set.
Pick automatic	PICKAUTO	Controls automatic windowing at the Select Objects prompt. *ON*: The *Window-Inside* and *Crossing Window* options of the *Select* command are chosen by default. This method is referred to as *Automatic Windowing*. *OFF*: You need to explicitly specify a selection method.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor ( ) . If you select an entity by clicking the *Pick Box* must touch or overlap the entity.

		The default size of the *Pick Box* is 3.
Pick drag	PICKDRAG	Controls the method of drawing a selection window. *ON*: Allows to define a selection window by dragging:press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. *OFF*: Define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. *ON*: Allows to first compose a selection set, then launch a modification command. *OFF*: You must first start the command, then compose the select entities.
Pick style	PICKSTYLE	Controls selection of associative hatch patterns and their boundaries. To select the hatch without its associated boundary, first set PICKSTYLE to 0 or 1. To select both the hatch and boundary, set PICKSTYLE to 2 or 3. To erase a boundary hatch pattern without erasing the boundary object, first set PICKSTYLE to 1. PICKSTYLE default setting is 1.

Selection Methods

Selection Methods

Before you start to use modification commands, you need to know how to compose a selection set. If the *PICKFIRST* variable is *ON* you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command.

Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click an entity.

  The entity highlights and grips show.

- Click to define the first corner of a selection window.

  Move the mouse to the left to define a *Crossing Window*:

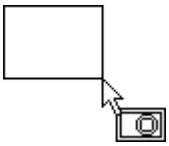
  

  All entities which overlap the window or are completely inside the window are added to the selection set.

  The selection window displays in dashed lines.

- Click to define the first corner of a selection window.

  Move the mouse to the right to define a *Window-Inside*:

  All entities which are completely inside the window are added to the selection set.

  The selection window displays in continuous lines.

- Press an hold the *Shift* key, then use one of the above methods to select entities that you want to *remove* from the selection set.

NOTE If the *PICKADD* variable is *OFF*, you cannot add entities to a selection set.

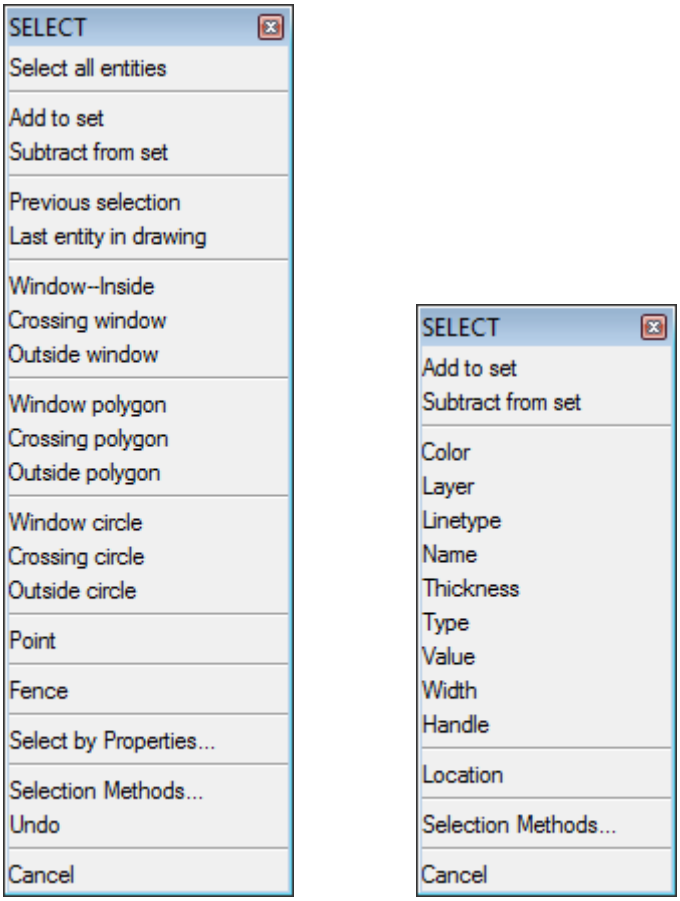
Composing a selection set afterwards (post-pick)

If you launch a modification command when no selection set is active, you are prompted to select entities. Bricscad provides a range of selection methods to let you compose your selection set easily.

The various selection methods are:

- *Add to set*: Selected entities are added to the selection set.
- *Subtract from set*: Selected entities are removed from the selection set.

- *Picking*: Place the pickbox over a part of the object and click.
- *Select all entities*: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- *Previous selection*: Re-use the previous selection set.
- *Last entity in drawing*: Select the most recent entity.
- *Window*: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a *rectangle*, *polygon* or a *circle*.
- *Fence*: All entities that cross a multi-segment line are selected.
- *Point*: Click a point to select all closed entities which enclose this point.
- *Select by Properties*: Displays the *Select by Properties* context menu to select entities by *Color*, *Layer*, *Linetype*, *Name*, *Thickness*, *Type*, *Value*, *Width* or *Handle*.
- *Undo*: Undoes the last selection action.
- *Location*: Displays the *Select Location* context menu.



Select Location and Select By Properties context menus

Rearranging Entities

Moving Entities

Moving Entities

You can move entities within the current drawing or from one drawing to another.

Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point0 and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

Some entities can be moved using grips. The grip you select depends on the type of entity. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.

Using the Move command

1. Do one of the following:

- Click the Move tool button () on the *Modify* toolbar.
- Choose Move in the *Modify* menu.
- Type *move* or *M* in the command bar.

The command bar reads: Select entities to move:

2. Select the entities, then right click or press Enter.

The command bar reads: Vector/<Base point>:

3. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Displacement point.

4. Do one of the following to specify the displacement point.

- Click the displacement point.
- Use Direct Distance Entry: type the displacement distance, then press Enter.

  The distance is measured in the cursor direction.

  Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection set is moved.

Command Options

Keyboard	Prompt Box	Description
V	Vector	You are prompted to enter the direction vector (x,y,z). E.g. a value of 5,4,2 moves the entities at a distance of 5 units in the x direction, 4 units in the y direction, and 2 units in the z direction. If you omit the z-coordinate it is assumed to be 0 (zero).
Esc	Cancel	Abort the Move command.

Moving entities using grips

1. Select the entity.

2. Click the grip.

   The entity moves with the cross hairs.

3. Click to relocate the entity.

NOTE The following entities can be moved using grips:

- lines, infinite lines and rays
- circles and circular arcs
- ellipses and elliptical arcs
- 3D solids
- text and multi-line text

Moving entities between drawings

Entities can be moved between drawings in one of three ways:

- Move command: allows to specify the base point in the selection set.
- Cut and Paste: allows to paste the selection using the coordinates of the source drawing.
- Cut and Paste Special: allows you to paste the selection as a block in the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

Using the Move command to move entities between drawings

1. Do one of the following:

- Click the Move tool button () on the *Modify* toolbar.
- Choose Move in the *Modify* menu.
- Type *move* or *M* in the command bar.

The command bar reads: Select entities to move:

2. Select the entities, then right click or press Enter.

The command bar reads: Vector/<Base point>:

3. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Displacement point.

4. Switch to the target drawing.

5. Specify the displacement point.

The selection set is moved from the source drawing into the target drawing.

Using Cut and Paste to move entities between drawings

1. In the source drawing, select the entities you want to move.

2. Do one of the following:

- Right click, then choose Cut ( ) in the context menu.
- Choose Cut ( ) in the *Edit* menu.
- Press *Ctrl + X* (press and hold the Ctrl key, then press X).

The entities are deleted in the source drawing and copied to the Clipboard.

3. Switch to the target drawing.

4. Do one of the following:

- Right click, then choose Paste ( ) in the context menu.
- Choose Paste ( ) in the *Edit* menu.
- Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:

- Specify the displacement point.
- Press Enter to paste the selection set using the coordinates of the source drawing.

NOTE If you choose Undo in the *Edit* menu in the source drawing, the deletion of the selection set is undone.

Paste a selection as a block in the target drawing

1. In the source drawing, select the entities you want to move.
2. Do one of the following:

- Right click, then choose Cut ( ) in the context menu.
- Choose Cut ( ) in the *Edit* menu.
- Press *Ctrl + X* (press and hold the Ctrl key, then press X).

The entities are deleted in the source drawing and copied to the Clipboard.

3. Switch to the target drawing.
4. Choose Paste Special in the *Edit* menu.
5. Choose *Paste As: Bricscad Block* in the *Paste Special* dialog window.
6. Press the *OK* button on the *Paste Special* dialog window.
7. Click the *Insert* button on the *Insert Block* dialog window to place the block.

You are prompted to specify the *Scaling* and *Rotation* of the block.

NOTE The origin of the current UCS in the source drawing will be the insertion point of

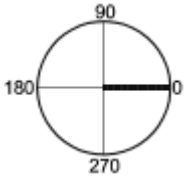
the block.

Rotating Entities

Rotating Entities

You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.



NOTES

- The *Angular Base* is controlled by the *ANGBASE* variable, which in turn refers to the current UCS.
- The *Angle Direction* is controlled by the *ANGDIR* variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

Rotating a selection set

1. Do one of the following:

- Click the Rotate tool button ( ) on the *Modify* toolbar.
- Choose Rotate in the *Modify* menu.
- Type *rotate* or *RO* in the command bar.

The command bar reads: Select entities to rotate:

2. Select the entities, then right click or press Enter.

The command bar reads: Rotation point:

3. Specify the rotation point.

The command bar reads: Base angle/<Rotation angle>:

4. Specify the rotation angle.

The selection set is rotated.

NOTE Use the *Center* option of the Array command if you want to keep the original entities.

Rotating a selection set in reference to a base angle

1. Do one of the following:

- Click the Rotate tool button ( ) on the *Modify* toolbar.
- Choose Rotate in the *Modify* menu.

- Type *rotate* or *RO* in the command bar.

The command bar reads: Select entities to rotate:

2. Select the entities, then right click or press Enter.

The command bar reads: Rotation point:

3. Specify the rotation point.

The command bar reads: Base angle/<Rotation angle>:

4. Do one of the following:

- Choose *Base angle* in the context menu.
- Type *B* in the command bar, then press Enter.

The command bar reads: Base angle <0>:

5. To specify the base angle do one of the following:

- Type the base angle in the command bar.
- Click the rotation point again, then click a second point.

The command bar reads: New Angle.

6. Do one of the following:

- Type the new angle in the command bar.
- Click a point to define the new angle.

Rotating in 3D

1. Do one of the following.

- Click the 3D Rotate tool button () on the *Modify* toolbar.
- Choose 3D Rotate in the *Modify* menu.
- Type *rotate3d* in the command bar.

The command bar reads: Select entities to rotate:

2. Select the entities, then right click or press Enter.

The command bar reads: Select axis by: Entity/Last/View/Xaxis/Yaxis/Z-axis/<2 points>:

3. Press Enter to define the rotation axis by specifying two points.

4. Specify the first rotation axis point.

5. Specify the second rotation axis point.

The command bar reads: Reference/<Rotation Angle>:

6. Specify the rotation angle.

The selection set is rotated clockwise, looking in the rotation axis direction.

Command Options

Keyboard	Prompt Box	Description

E	Entity	Select a line or polyline segment to define to rotation axis. The direction of the line or segment defines the rotation axis dimension.
L	Last	Re-use the rotation axis used in the previous 3D Rotate.
V	View	Rotate about an axis which is perpendicular to the current view. You are prompted to specify the location of the rotation axis.
X	Xaxis	Rotate about the X-axis of the current UCS.
Y	Yaxis	Rotate about the Y-axis of the current UCS.
Z	Zaxis	Rotate about the Z-axis of the current UCS.
R	Reference	Specify a reference angle.
Esc	Cancel	Abort the 3D Rotate command.

Adjusting the Draw Order

If new entities overlap existing entities, they will display and print on top of the previously drawn entities. The Draw Order tool can change the order in which overlapping entities are displayed and printed. You can move entities to the front, back, or on top or below of another entity.

Using Draw Order

1. Do one of the following:

- Click the Draw Order tool button () on the Draworder toolbar.
- Choose Draw Order in the *Tools* menu.
- Type *draworder* in the command bar.

The command bar reads: Select entities to change the draw order:

2. Select the entities, then right click or press enter.

The command bar reads: Change draw order: Above/Under/Clear/Front/<Back>:

3. Choose the appropriate draw order tool option.

4. If you have chosen *Above* or *Under*, the command bar reads: Select reference entity:

Identify the reference entity.

Command Options

Keyboard	Prompt Box	Description
A	Above	Places the selection set above the reference entity.
U	Under	Places the selection set under the reference entity.
C	Clear	Clears the draw order rearrangement for the selection set.
F	Front	Places the selection set on top.
B	Back	Moves the selection set to the back.

Using Draw Order tools

1. Select the entities you want to change the draw order of.
2. Do one of the following.

- Click the appropriate tool button on the *Draw Order* toolbar.
- Right click, then choose *Draw Orde*r in the context menu and select the appropriate tool.

Tool Button	Tool Name	Description

	Bring to front	Places the selection set on top.
	Move Forward	Places the selection set above the reference entity. (*)
	Move Backward	Places the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.

(*) You are prompted to select the reference entity.

Aligning Entities

Aligning Entities

The Align command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary you can even scale the selection to fit in its new location.

The Align command can be used both in a 2D and 3D environment.

Aligning an entity in 2D

1. Do one of the following:

- Click the Align tool button ( ) on the *Modify* toolbar.
- Choose Align in the *Modify* menu.
- Type *align* in the command bar, then press Enter.

The command bar reads: Select entities:

2. Select the entities, then right click or press Enter.

The command bar reads: Specify first source point:

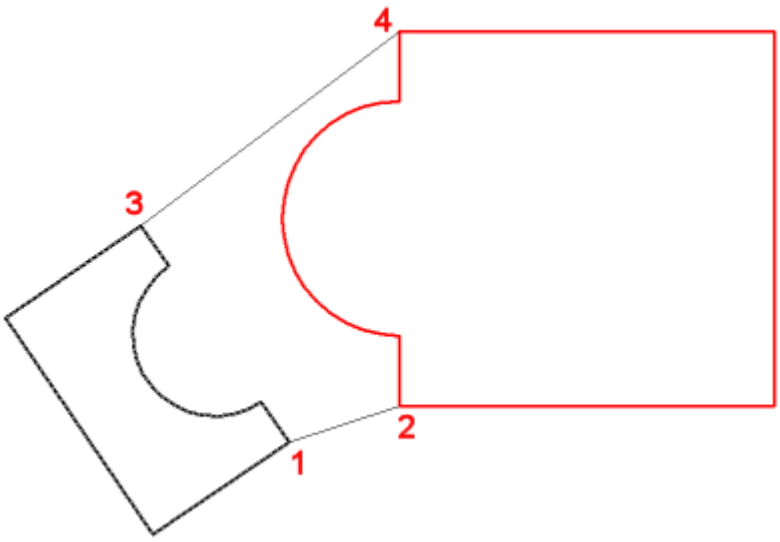
3. Snap to the first source point in the selection set (1).

The command bar reads: Specify first destination point:

4. Snap to the first destination point on the reference entity (2).

A witness line is drawn between the source point and the target point.

The command bar reads: Specify the second source point:



5. Snap to the second source point in the selection set (3).

The command bar reads: Specify second destination point:

6. Snap to the second destination point on the reference entity (4).

A witness line is drawn between the source point and the target point.

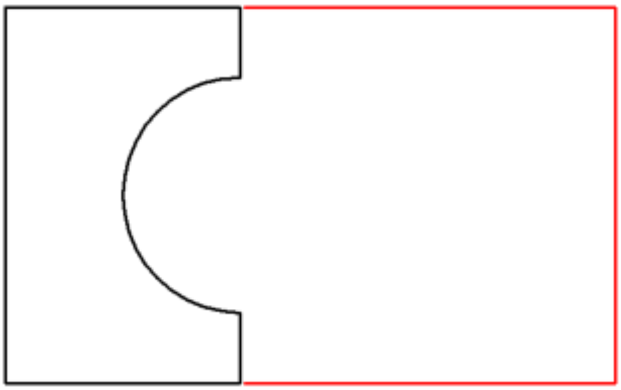
The command bar reads: Specify the third source point:

7. Right click to skip the third source point.

The command bar reads: Scale objects based on alignment points [Yes/No] <No>:

8. Do one of the following:

- Press Enter if you don't wan to scale the selection set.
- Type Y and press Enter to scale the selection set.



Result of the Align procedure with scaled selection set.

Aligning an entity in 3D

1. Do one of the following:

- Click the Align tool button ( ) on the *Modify* toolbar.
- Choose Align in the *Modify* menu.
- Type *align* in the command bar, then press Enter.

The command bar reads: Select entities:

2. Select the entities, then right click or press Enter.

The command bar reads: Specify first source point:

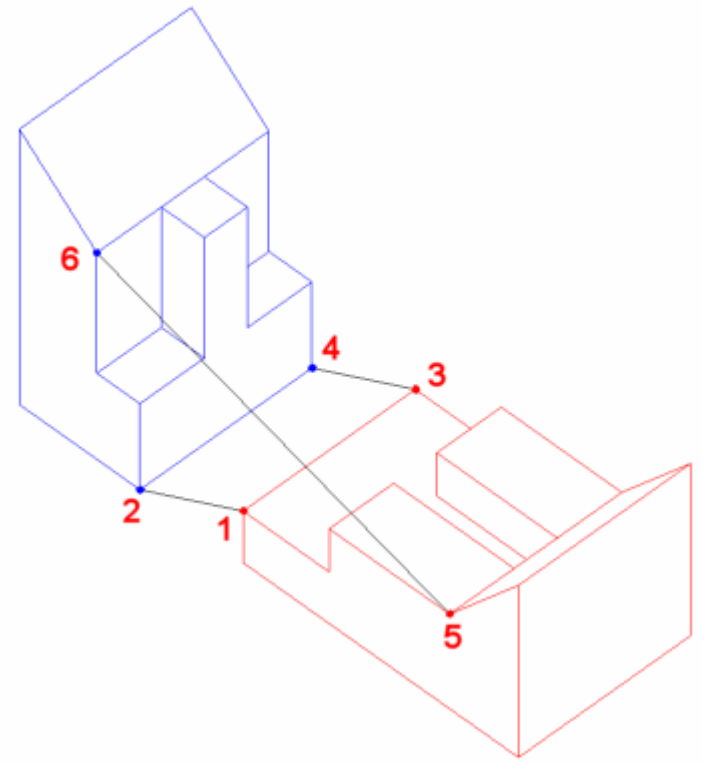
3. Snap to the first source point in the selection set (1).

The command bar reads: Specify first destination point:

4. Snap to the first destination point on the reference entity (2).

A witness line is drawn between the source point and the target point.

The command bar reads: Specify the second source point:



5. Snap to the second source point in the selection set (3).

The command bar reads: Specify second destination point:

6. Snap to the second destination point on the reference entity (4).

A witness line is drawn between the source point and the target point.

The command bar reads: Specify the third source point:

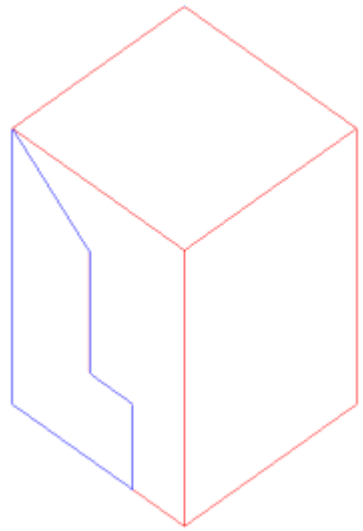
7. Snap to the third source point in the selection set (5).

The command bar reads: Specify third destination point:

8. Snap to the third destination point on the reference entity (6).

A witness line is drawn between the source point and the target point.

The entity is moved.



Changing Entities

The Change tool lets you change the insertion point and orientation of texts and block inserts.

The *Properties* option of the command can also change the layer, line type, line type scale, line weight, etc. but he Bricscad Properties bar is much more versatile to do such things.

To change text entities

1. Do one of the following

- Click the Change tool button () on the *Modify* toolbar.
- Choose Change in the *Modify* menu
- Type *change* in the command bar, then press Enter.

The command bar reads: Select entities to change:

2. Select the text entities, then right click or press Enter.

The command bar reads: Change: Entities/Properties/<Change point>:

3. Right click or press Enter to accept the default option.

New point for text, or Enter for no change:

The first text entity is attached to the cursor.

4. Specify the new insertion point for the text or right click to accept the current location.

The command bar reads: New text style <current style>:

5. Type a new text style and press Enter or right click to keep the current style.

The command bar reads: New height <current height>:

6. Type a new height and press Enter or right click to keep the current height.

The command bar reads: New rotation angle <current angle>:

7. Type a new angle and press Enter or right click to keep the current angle.

The command bar reads: New text <current text>:

8. Type the new text and press Enter or right click to keep the current text.

9. (option) If multiple text entities were selected in step 2, steps 3 through 8 are repeated for each text.

NOTE The Change tool cannot change Multi-Line text (Mtext) entities.

Copying Entities

Copying Entities in a Drawing

You can duplicate entities within the current drawing. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) for the copy. You can also make multiple copies or copy the selection set using a direction vector.

Making a single copy

1. Do one of the following:

- Click the Copy tool button () on the *Modify* toolbar.
- Choose Copy in the *Modify* menu.
- Type *copy* or *CO* in the command bar.

The command bar reads: Select entities to copy:

2. Select the entities, then right click or press Enter.

The command bar reads: Multiple/Vector/<Base Point>:

3. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Displacement point.

4. Do one of the following to specify the displacement point.

- Click the displacement point.
- Use Direct Distance Entry: type the displacement distance, then press Enter.

  The distance is measured in the cursor direction.

  Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection set is copied.

Command Options

Keyboard	Prompt Box	Description
M	Multiple Copies	Creates multiple copies of the selection set.
V	Vector	You are prompted to enter the direction vector (x,y,z). E.g. a value of 5,4,2 moves the entities at a distance of 5 units in the x direction, 4 units in the y direction, and 2 units in the z direction. If you omit the z-coordinate it is assumed to be 0 (zero).
Esc	Cancel	Abort the Move command.

Making multiple copies

1. Do one of the following:

- Click the Copy tool button () on the *Modify* toolbar.
- Choose Copy in the *Modify* menu.
- Type *copy* or *CO* in the command bar.

The command bar reads: Select entities to copy:

2. Select the entities, then right click or press Enter.

The command bar reads: Multiple/Vector/<Base Point>:

3. Choose *Multiple Copies* in the prompt box or type *M*, then press Enter.

The command bar reads: Vector/<Base Point>:

4. Do one of the following to specify the displacement point.

- Click the displacement point.
- Type the displacement distance, then press Enter.
  The distance is measured in the cursor direction.
  Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection set is copied.

5. Repeat step 4 to place additional copies.
6. Right click or press Enter to stop.

Copying Entities Between Drawings

Copying Entities Between Drawings

Entities can be copied between drawings in one of three ways:

- Copy command: allows to specify the base point in the selection set.
- Copy and Paste: allows to paste the selection using the coordinates of the source drawing.
- Copy and Paste Special: allows you to paste the selection as a block in the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

Using the Copy command to copy entities between drawings

1. Do one of the following:

- Click the Copy tool button () on the *Modify* toolbar.
- Choose Copy in the *Modify* menu.
- Type *Copy* or *CO* in the command bar.

The command bar reads: Select entities to Copy:

2. Select the entities, then right click or press Enter.

The command bar reads: Multiple/Vector/<Base point>:

3. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Displacement point.

4. Switch to the target drawing.

5. Specify the displacement point.

The selection set is Copied from the source drawing into the target drawing.

Using Copy and Paste to Copy entities between drawings

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:

- Right click, then choose Copy () in the context menu.
- Choose Copy () in the *Edit* menu.
- Press *Ctrl + C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.

3. Switch to the target drawing.
4. Do one of the following:

- Right click, then choose Paste () in the context menu.
- Choose Paste () in the *Edit* menu.
- Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:

- Specify the displacement point.
- Press Enter to paste the selection set using the coordinates of the source drawing.

Paste a selection as a block in the target drawing

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:

- Right click, then choose Copy () in the context menu.
- Choose Copy () in the *Edit* menu.
- Press *Ctrl + C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.

3. Switch to the target drawing.
4. Choose Paste Special in the *Edit* menu.
5. Choose *Paste As: Bricscad Block* in the *Paste Special* dialog window.
6. Press the *OK* button on the *Paste Special* dialog window.
7. Click the *Insert* button on the *Insert Block* dialog window to place the block.

You are prompted to specify the *Scaling* and *Rotation* of the block.

NOTE The origin of the current UCS in the source drawing will be the insertion point of the block.

Making Parallel Copies

The parallel command creates a copy of linear entities and align them parallel to the original entities at a specified distance. You can make parallel entities using arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

NOTE Depending on the specified distance and the shape of the selected entity in some cases it might be impossible to create a parallel copy.

Making a parallel copy at a specified distance

1. Do one of the following:

- Click the Parallel tool button () on the *Modify* toolbar.
- Choose Parallel in the *Modify* menu.
- Type *parallel* in the command bar.

The command bar reads: Parallel: Through point/<Distance> <current distance>:

2. Do one of the following:

- Press Enter to accept the current distance.
- Type a new distance in the command bar.
- Define a new distance by specifying two points.

The command bar reads: Select Entity.

3. Select the entity.

The command bar reads: Both sides/<Side for parallel copy>:

4. Click the side for the parallel copy.

The parallel copy is created.

The command bar reads: Select Entity.

5. Do one of the following:

- Repeat steps 3 and 4 to create more parallel copies.
- Right click or press Enter to stop.

Command Options

Keyboard	Prompt Box	Description
T	Trough Point	Creates a parallel copy through a specific point.
B	Both Sides	Creates a parallel copy on both sides of the selected entity.
L	Last	Selects the last entity again.

Esc	Cancel	Aborts the parallel command.

Making a parallel copy through a point

1. Do one of the following:

- Click the Parallel tool button () on the *Modify* toolbar.
- Choose Parallel in the *Modify* menu.
- Type *parallel* in the command bar.

The command bar reads: Parallel: Through point/<Distance> <current distance>:

2. Choose *Through point* in the context menu or type *T*, then press Enter.

The command bar reads: Select entity:

3. Select the entity.

The command bar reads: Through point:

4. Specify the point to create the parallel copy through.

The parallel copy is created.

The command bar reads: Select entity:

5. Do one of the following.

- Repeat steps 3 and 4 to create ;ore parallel copies.
- Right click or press Enter to stop.

Mirroring Entities

Mirroring Entities

You can create a mirror image of a selection set. The selection is mirrored about a mirro line, which you define by specifying two points. You can choose to either delete or retain the original entities.

You can also create a mirror image of selected entities in three-dimensional space. In this case the selection is mirrored about a mirror plan, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the xy, yz, or xz plane of the current UCS or aligning the plane with the current view. You can choose to either delete or retain the original entities.

NOTE Whether text is mirrored or not by the Mirror command is controlled by the *Mirror Text* (MIRRTEXT) setting.

Mirroring entities about a line

1. Do one of the following:

- Click the Mirror tool button () on the *Modify* toolbar.
- Choose Mirror in the *Modify* menu.
- Type *mirror* or *MI* in the command bar.

The command bar reads: Select entities to mirror:

2. Select the entities then right click or press Enter.

The command bar reads: Start of mirror line.

3. Specify the start point of the mirror line.

4. Specify the endpoint of the mirror line.

The command bar reads: Delete the original entities? <N>

5. Do one of the following:

- Press Enter to keep the original entities.
- Type *Y*, then press Enter to delete the original entities.
- Choose *Yes-Delete entities* in the context menu.

Mirroring entities about a three-dimensional plane

1. Do one of the following:

- Click the 3D Mirror tool button () on the *Modify* toolbar.
- Choose 3D Mirror in the *Modify* menu.
- Type *mirror3d* in the command bar.

The command bar reads: Select entities:

2. Select the entities then right click or press Enter.

The command bar reads: Define mirror plane by: Entity/Last/View/Zaxis/XY/YZ/ZX/<3points>:

3. Specify the first point of the mirror plane.

4. Specify the second point of the mirror plane.

5. Specify the third point of the mirror plane.

The command bar reads: Delete the original entities? <N>

6. Do one of the following:

- Press Enter to keep the original entities.
- Type *Y*, then press Enter to delete the original entities.
- Choose *Yes-Delete entities* in the context menu.

Command Options

Keyboard	Prompt Box	Description
E	Entity	Select a planar entity to define the mirror plane.
L	Last	Re-uses the last mirror plane.
V	View	
Z	Z axis	Define the mirror plane by two points: the mirror plane passes through the first point and is perpendicular to the line defined by the two points.
XY	X-Y plane	Define the mirror plane parallel to the XY-plane of the current UCS.
YZ	Y-Z plane	Define the mirror plane parallel to the YZ-plane of the current UCS.
ZX	Z-X plane	Define the mirror plane parallel to the ZX-plane of the current UCS.
Enter	3 points	Define the mirror plane by three points (default option).
Esc	Cancel	Abort the 3D Mirror command.

Arraying Entities

Arraying Entities

The Array command copies a selection set in a rectangular or polar (circular) pattern. For a rectangular array you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set about the center point or to keep its original orientation.

Creating a rectangular array

1. Do one of the following:

- Click the 2D Array tool button () on the *Modify* toolbar.
- Choose 2D Array in the *Modify* menu.
- Type *AR* or *array* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Polar/<Rectangular>:

3. Choose *Rectangular* in the prompt box or type *R* in the command bar and press Enter.

The command bar reads: Number of rows in the array <1>:

4. Type the number of rows in the command bar, then press Enter.

The command bar reads: Number of columns in the array <1>:

5. Type the number of columns in the command bar, then press Enter.

The command bar reads: Vertical distance between rows, or spacing rectangle:

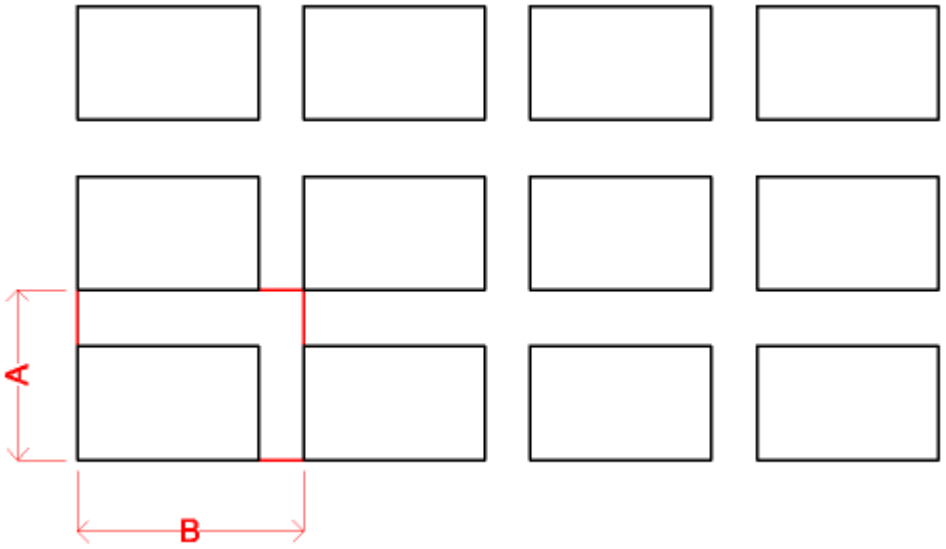
6. Type the distance between the rows of the array and press Enter.

The command bar reads: Horizontal distance between columns:

7. To define the distance between the columns of the array, do one of the following:

- Type the distance in the command bar and press Enter.
- Specify the distance by clicking two points.

The array is created.



Row spacing (A) and column spacing (B) can be defined by the spacing rectangle (red).

NOTE Positive values in steps 4 and 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

Creating a polar array

1. Do one of the following:

- Click the 2D Array tool button ( ) on the *Modify* toolbar.
- Choose 2D Array in the *Modify* menu.
- Type *AR* or *array* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Polar/<Rectangular>:

3. Choose *Polar* in the prompt box or type *P* in the command bar and press Enter.

The command bar reads: Base/Center of polar array:

4. Specify the center point of the polar array.

The command bar reads: ENTER to specify angle between items/<Number of items to array>:

5. Type the number of items in the array, then press Enter.

The command bar reads: Angle to array (+ for ccw, - for cw) <360>:

6. Do one of the following:

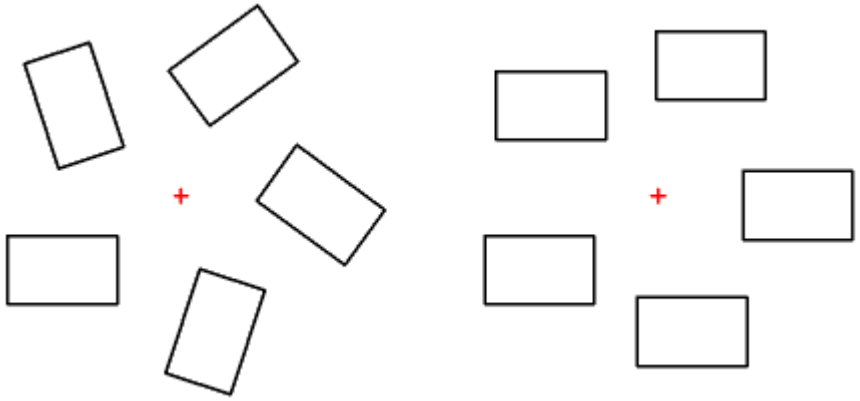
- Type the angle to array, then press Enter.
- Press Enter to create a 360° array.

The command bar reads: Rotate entities around the array? No/<Yes>

7. Do one of the following:

- Press Enter to rotate the entities about the center point of the array.
- Type N and press Enter to keep the original orientation of the entities.

The array is created.



Entities rotated (left) or not (right) about the center point of the array.

Creating a polar array with base point

1. Do one of the following:

- Click the 2D Array tool button ( ) on the *Modify* toolbar.
- Choose 2D Array in the *Modify* menu.
- Type *AR* or *array* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Polar/<Rectangular>:

3. Do one of the following:

- Type *P* in the command bar, then press Enter.
- Choose *Polar* in the prompt box.

The command bar reads: Base/Center of polar array:

4. Choose *Base* in the prompt box or type *B* and press Enter.

The command bar reads: Base of polar array entities.

5. Specify the base point for the rotation of the selection set.

The command bar reads: Center point of array:

6. Specify the center point of the array.

The command bar reads: ENTER to specify angle between items/<Number of items to array>:

7. Type the number of items in the array, then press Enter.

The command bar reads: Angle to array (+ for ccw, - for cw) <360>:

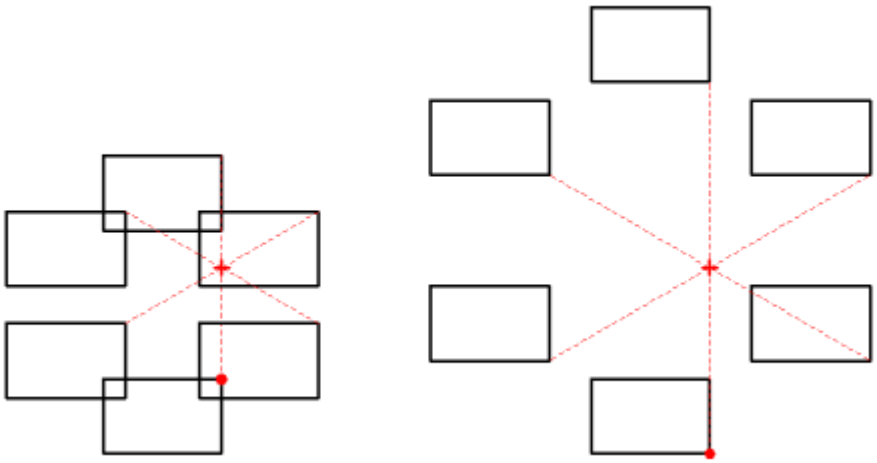
8. Do one of the following:

- Type the angle to array the press Enter.
- Press Enter to create a 360° array.

The command bar reads: Rotate entities around the array? No/<Yes>

9. Type N and press Enter to keep the original orientation of the entities.

The array is created.



Polar array with base point (dot) and center point (cross).

NOTE If you choose to rotate the entities about the center point in the final step of the array procedure, the selection of a base point does not influence the result.

Specify the angle between the items when creating a polar array

1. Do one of the following:

- Click the 2D Array tool button ( ) on the *Modify* toolbar.
- Choose 2D Array in the *Modify* menu.
- Type *AR* or *array* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Polar/<Rectangular>:

3. Choose *Polar* in the prompt box or type *P* in the command bar, then press Enter.

The command bar reads: Base/Center of polar array:

4. Specify the center of the polar array.

The command bar reads: ENTER to specify angle between items/<Number of items to array>:

5. Press Enter.

The command bar reads: Angle to array (+ for ccw, - for cw) <360>:

6. Specify the angle to array, then press Enter.

The command bar reads: Angle between items:

7. Specify the angle between the items and press Enter.

The command bar reads: Rotate entities around the array? No/<Yes>:

8. Do one of the following:

- Press Enter to rotate the entities.
- Type N, then press Enter to keep the original orientation of the entities.

The array is created.

NOTE This procedure can also be used to rotate a selection set and keep the original.

Arraying Entities in 3D

Arraying Entities in 3D

The 3D Array command creates multiple copies a selection set in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns, and levels) pattern or a two-dimensional polar (circular) pattern. The polar pattern is created by copying entities about a specified axis. You can choose to rotate the selection set about rotation axis or or to keep its original orientation.

Creating a 3D rectangular array

1. Do one of the following:

- Click the 3D Array tool button () on the *Modify* toolbar.
- Choose 3D Array in the *Modify* menu.
- Type *3darray* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Type of array: Polar/<Rectangular>:

3. Choose *Rectangular* in the prompt box or type *R* in the command bar and press Enter.

The command bar reads: Number of rows in the array <1>:

4. Type the number of rows in the command bar, then press Enter.

The command bar reads: Number of columns <1>:

5. Type the number of columns in the command bar, then press Enter.

The command bar reads: Number of levels <1>:

6. Type the number of levels in de command bar, then press Enter.

The command bar reads: Vertical distance between rows:

7. To define the distance between rows of the array, do one of the following:

- Type the distance in the command bar and press Enter.
- Specify the distance by clicking two points.

The command bar reads: Horizontal distance between columns:

8. To define the distance between the columns of the array, do one of the following:

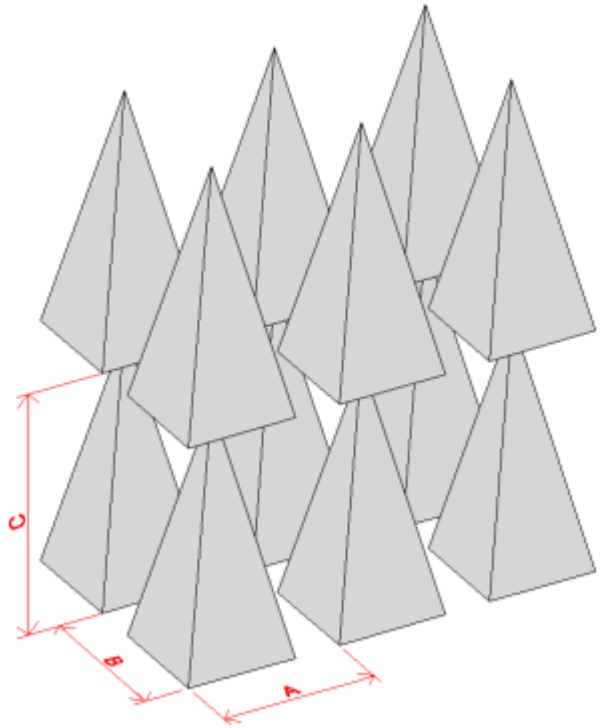
- Type the distance in the command bar and press Enter.
- Specify the distance by clicking two points.

The command bar reads: Depth between levels:

9. To define the distance between the levels of the array, do one of the following:

- Type the distance in the command bar and press Enter.
- Specify the distance by clicking two points.

The array is created.



Rectangular 3D Array:row spacing A, column spacing B, level spacing C

NOTE Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

Creating a 3D polar array

1. Do one of the following:

- Click the 3D Array tool button ( ) on the *Modify* toolbar.
- Choose 3D Array in the *Modify* menu.
- Type *3darray* in the command bar, then press Enter.

The command bar reads: Select entities to array:

2. Select the entities, then right click or press Enter.

The command bar reads: Type of array: Polar/<Rectangular>:

3. Choose *Polar* in the prompt box or type *P* in the command bar and press Enter.

The command bar reads: ENTER to specify angle between items/<Number of items to array>:

4. Type the number of items you want in the array, then press Enter.

The command bar reads: Angle to array (+ for ccw, - for cw) <360>:

5. Do one of the following:

- Type the angle to array, then press Enter.
- Press Enter to create a 360° array.

The command bar reads: Rotate entities around the array? No/<Yes>

6. Do one of the following:

- Press Enter to rotate the entities about the array axis.
- Type N and press Enter to keep the original orientation of the entities.

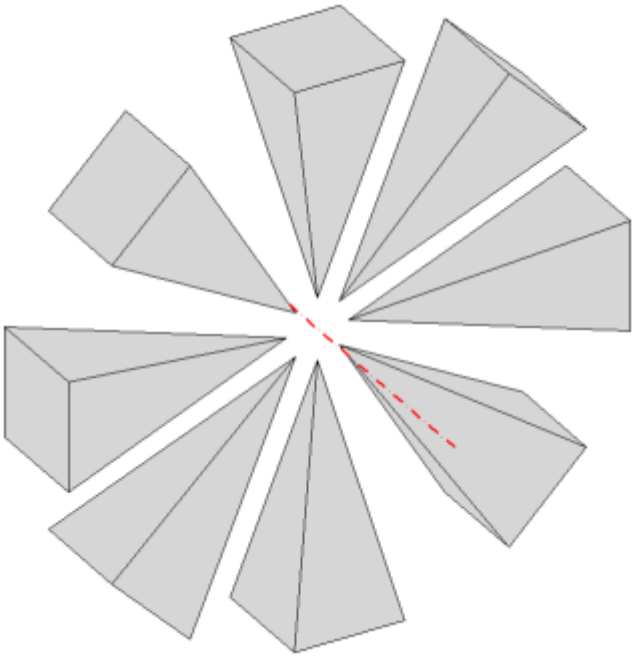
The command bar reads: Center of polar array.

7. Specify the first point of the array axis.

The command bar reads: Specify second point along central axis of array:

8. Specify the second point of the array axis.

The array is created.



Polar 3D array about a horizontal axis (red).

Resizing Entities

Extending Entities

The Extend command lets you extend entities to a boundary, which is defined by one or more other entities.

If the *Edge Mode* setting (EDGEMODE) is *On*, you can extend entities to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the *Projection Mode* setting (PROJMODE) lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When extending entities, you first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

To extend entities

1. Do one of the following:

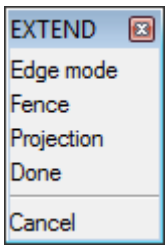
- Click the Extend tool button () on the *Extend/Stretch* flyout of *Modify* toolbar.
- Choose Extend in the *Modify* menu.
- Type *extend* or *EX* in the command bar, then press Enter.

The command bar reads: Select boundary entities for extend <ENTER to select all>:

2. Select the boundary entities, then press Enter or right click.

The command bar reads: Edge mode/Fence/Projection/<Select entity to extend>:

A prompt box displays.



3. Click the entity you want to extend near the end that can make the extension.

The entity is extended.

4. Repeat step 3 to extend more entities.

5. Right click to conclude the Extend command.

Command Options

Keyboard	Prompt Box	Description

E	Edge mode	Adjusts the *Edge Mode* setting. The options are: *Extend* and *No Extend*.
F	Fence	Lets you select the entities to extend using a fence: all entities that cross the fence line are extended.
P	Projection	Adjusts the *Projection Mode* setting. The options are: *No Projection*, *xy plane of UCS* and *Current view*.
Enter	Done	Concludes the Extend command.
Esc	Cancel	Aborts the Extend command.

NOTES

- Entities which are selected when you launch the Extend command will be used as boundary entities.
- If you select an entity near the end that cannot make an extension to on of the boundary entities, the Extend command is aborted.

Changing the length of an entity

Changing the length of an entity

The *Edit Length* (EDITLEN) command lets you change the length of lines, open polylines and arcs. You can also modify the included angle of arcs.

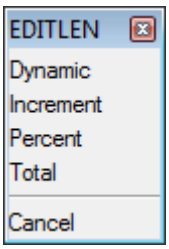
Change the length of an entity dynamically

1. Do one of the following:

- Click the Edit Length tool button ( ) on the *Extend/Stretch* flyout of *Modify* toolbar.
- Choose Edit Length in the *Modify* menu.
- Type *editlen* in the command bar, then press Enter.

  The command bar reads: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

  A prompt box displays:



2. (option) Click an entity.

The current length of the selected entity displays in the command bar.

In case an arc is selected, the current length and the included angle display in the command bar.

3. Do on of the following:

- Choose Dynamic in the prompt box.
- Type DY in the command bar, then press Enter.

  The command bar reads: Mode/<Select entity to change>:

4. Click the entity near the end you want to change.

The length of the entity changes dynamically.

5. Click to define the new length.

Command Options

Keyboard	Prompt Box	Description
DY	Dynamic	Changes the length dynamically.
I	Increment	Lengthens or shortens the length of an entity by the specified increment.
P	Percent	Changes the length of an entity by the specified percent.

T	Total	Changes the total length of an entity to the specified length.
Esc	Cancel	Aborts the Edit Length command.

Modify the included angle of an arc

1. Do one of the following:

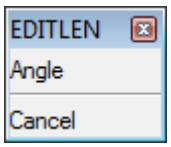
- Click the Edit Length tool button ( ) on the *Extend/Stretch* flyout of *Modify* toolbar.
- Choose Edit Length in the *Modify* menu.
- Type *editlen* in the command bar, then press Enter.

The command bar reads: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

2. Choose *Total* in the prompt box of type *T* and press Enter.

The command bar reads: Angle/<Enter total Length (0.00)>:

A prompt box displays:



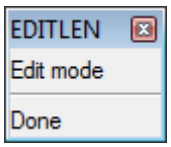
3. Choose *Angle* in the prompt box or type *A* and press Enter.

The command bar reads: Enter total angle <00° 0' 0">:

4. Type the new angle in the command bar and press Enter.

The command bar reads: Mode/<Select entity to change>:

A prompt box displays:

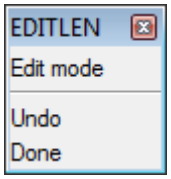


5. Click the arc at the end you want to lengthen or shorten.

The arc is modified.

The command bar reads: Mode/Undo/<Select entity to change>:

A prompt box displays:



6. Do one of the following:

- Select another arc.
- Choose *Undo* in the prompt box or type *U* and press Enter to undo the previous action.
- Choose *Edit mode* in the prompt box or type *M* and press Enter to choose a different Edit Length command option.

- Choose *Done* in the prompt box to conclude the Edit Length command.

Stretching Entities

Stretching Entities

The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.

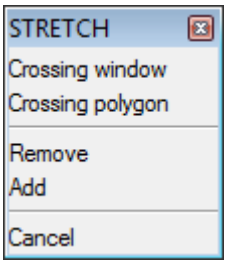
To stretch entities

1. Do one of the following:

- Click the Stretch tool button () on the *Extend/Stretch* flyout of the *Modify* toolbar.
- Choose Stretch in the *Modify* menu.
- Type s*tretch* or *S* in the command bar and press Enter.

  The command bar reads: Select entities to stretch by crossing-window or crossing-polygon:

  A prompt box displays:



2. (option) Choose *Crossing window* in the *Stretch* prompt box, then define the stretch area by a rectangular window.

3. (option) Choose *Crossing polygon* in the *Stretch* prompt box, then define the stretch area by a polygon.

4. (option) Repeat steps 2 and 3 to expand the stretch area.

5. (option) Choose *Remove* in the *Stretch* prompt box to select entities in the stretch area that must not be stretched.

6. (option) Choose *Add* in the *Stretch* prompt box to add previously remove entities to the selection of entities that must be stretched.

7. Right click to conclude the selection of entities.

The command bar reads: Base point of displacement.

8. Specify the base point.

The selection stretches dynamically.

The command bar reads: Second point of displacement.

9. Do one of the following to specify the second displacement point.

- Click the displacement point.

- Use Direct Distance Entry: type the displacement distance, then press Enter.

  The distance is measured in the cursor direction.

  Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection is stretched.

NOTES To add or remove entities from the selection set in steps 5 and 6 you can use any selection method: picking, window inside or crossing window.

Stretching entities using grips

1. Click the entity you want to stretch.

   The entity grips display.

2. Click a grip to activate it.

   The grip is attached to the drawing cursor.

3. Click to relocate the grip.

   The grip is released from the drawing cursor.

NOTES

- Use Ortho Mode, Polar Tracking or Snap Tracking to constrain the movement of the drawing cursor.
- If you select two (or more) entities with coinciding grips, the shared grips move simultaneously.

Trimming Entities

The Trim command lets you clip or trim entities by cutting entities.

If the *Edge Mode* setting (EDGEMODE) is *On*, you can trim entities by an implied edge of the cutting entities.

If the cutting entity is not in the same plane as the entity you want to trim, the *Projection Mode* setting (PROJMODE) lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

To trim entities

1. Do one of the following:

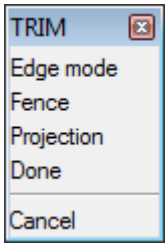
- Click the Trim tool button () on the *Modify* toolbar.
- Choose Trim in the *Modify* menu.
- Type *trim* or *TR* in the command bar, then press Enter.

The command bar reads: Select cutting entities for trim <ENTER to select all>:

2. Select the cutting entities, then press Enter or right click.

The command bar reads: Edge mode/Fence/Projection/<Select entity to trim>:

A prompt box displays.



3. Click the entity you want to trim, the part of the entity that you click will be removed.

The entity is trimmed.

4. Repeat step 3 to trim more entities.

5. Right click to conclude the Trim command.

Command Options

Keyboard	Prompt Box	Description
E	Edge mode	Adjusts the *Edge Mode* setting. The options are: *Extend* and *No Extend*.
F	Fence	Lets you select the entities to trim using a fence: all entities that

		cross the fence line are trimmed.
P	Projection	Adjusts the *Projection Mode* setting. The options are: *No Projection*, *xy plane of UCS* and *Current view*.
Enter	Done	Concludes the Trim command.
Esc	Cancel	Aborts the Trim command.

NOTES Entities which are selected when you launch the Trim command will be used as cutting entities.

Scaling Entities

The Scale command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the command bar. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing.

To scale a selection set

1. Do one of the following:

- Click the Scale tool button () on the *Modify* toolbar
- Choose Scale in the *Modify* menu.
- Type *scale* or *SC* in the command bar.

The command bar reads: Select entities to scale:

2. Select the entities, then right click or press Enter.

The command bar reads: Base point:

3. Specify the base point for the scaling.

The selection set scales dynamically.

The command bar reads: Base scale/<Scale Factor>:

4. Type the scale factor in the command bar and press Enter.

The selection set is scaled.

NOTE The length of the vector between the base point and the current cursor position is used as the dynamic scale factor in step 3.

Scaling an entity using a base scale

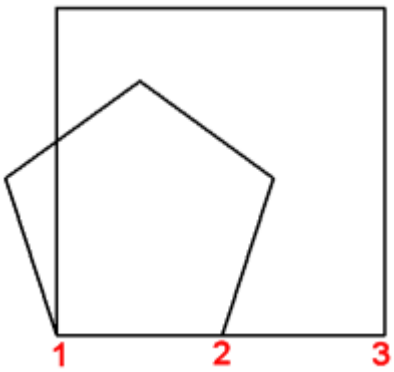
1. Select the entity.
2. Do one of the following:

- Click the Scale tool button () on the *Modify* toolbar
- Choose Scale in the *Modify* menu.
- Type *scale* or *SC* in the command bar.

The command bar reads: Base point:

3. Specify the base point (1).

The command bar reads: Base Scale/<Scale Factor>:



4. Choose *Base* in the prompt box of type *B* and press Enter.

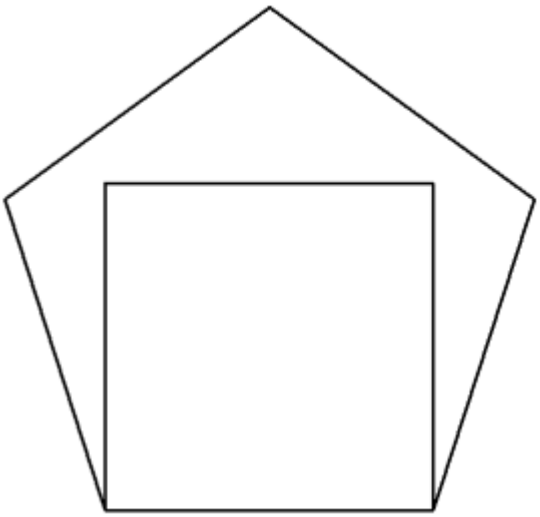
The command bar reads: Base scale <1>:

5. Click point 1, then point 2 to define the base scale.

The pentagon scales dynamically.

6. Click point 3.

The edge of the pentagon now equals the side of the square.



Breaking and Joining Entities

Breaking Entities

The Break command remove a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

To break an entity

1. Do one of the following:

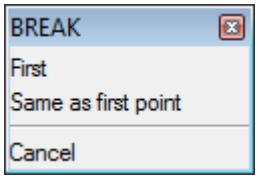
- Click the Break tool button () on the *Modify* toolbar.
- Choose Break in the *Modify* menu.
- Type *break* or *BR* in the command bar, then press Enter.

The command bar reads: Select entity to break:

2. Click the entity you want to break.

The command bar reads: First break point/<Second break point>:

A prompt box displays:



3. Click a second point on the entity.

The entity is broken. The portion between the two break points is removed.

Command Options

Keyboard	Prompt Box	Description
F	First	Lets you specify the first break point.
@	Same as first point	Breaks the entity into two parts, without removing a portion.
Esc	Cancel	Aborts the Break command.

Joining Entities

You can join two entities into a single entity. You can join either two lines or two arcs. Lines must be parallel, while arcs must share the same center point and radius.

When you join two lines, the farthest endpoints remain at their existing locations; the program draws a new line between these points.

Arcs are joined counterclockwise, therefore the result depends on the selecting order.

To join two lines

1. Do one of the following:

- Click the Join tool button ( ) on the *Modify* toolbar.
- Choose Join in the *Modify* menu.
- Type *join* in the command bar, then press Enter.

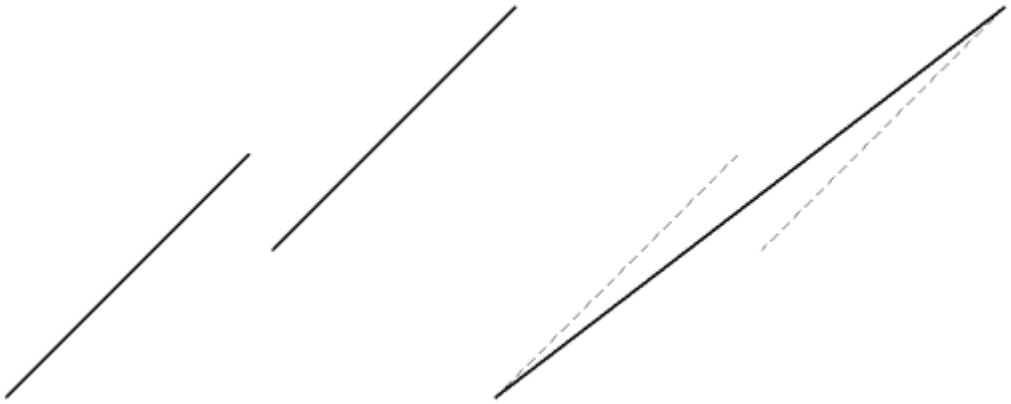
The command bar reads: Select arc or line for joining:

2. Select the first line.

The command bar reads: Select lines for attempted join:

3. Select the second line and press Enter.

The lines are joined.



Joining two lines: start situation (left) and result (right).

To join two arcs

1. Do one of the following:

- Click the Join tool button ( ) on the *Modify* toolbar.
- Choose Join in the *Modify* menu.
- Type *join* in the command bar, then press Enter.

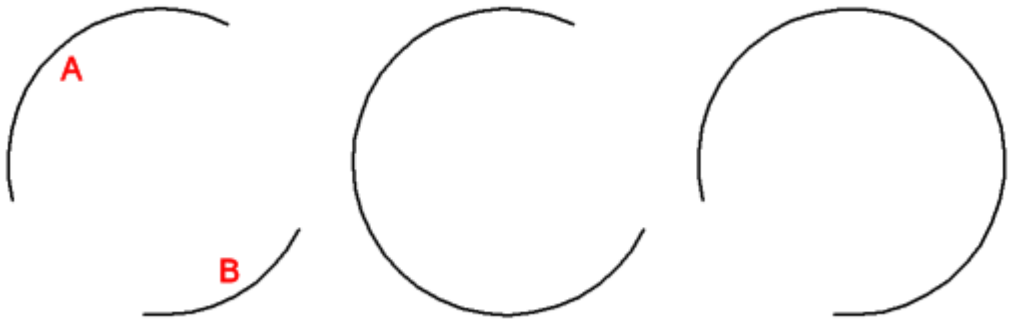
The command bar reads: Select arc or line for joining:

2. Select the first arc.

The command bar reads: Select arcs for counterclockwise join:

3. Select the second arc and press Enter.

The arcs are joined.



Joining two arcs: start situation (left) and result when selecting A first (middle) or B first (right).

NOTE If more arcs share the same center point and radius, you can select the in step 3. Press Enter after selecting the last arc.

Chamfering and Filleting

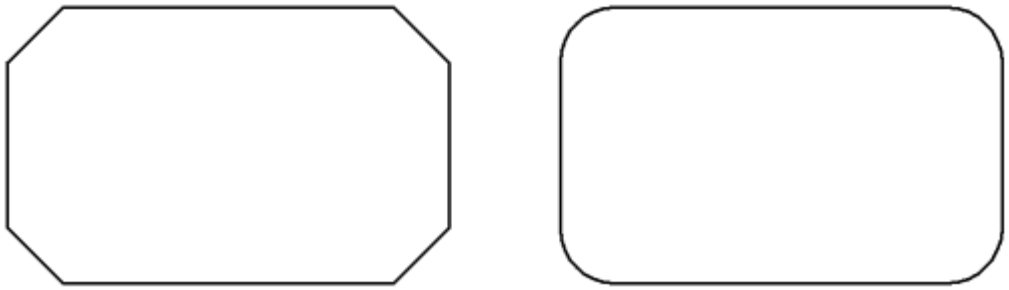
Overview

A chamfer connects two nonparallel entities with a line to create a beveled edge.

A fillet connects two entities with an arc of a specified radius to create a rounded edge.

If both entities you are working with are on the same layer, the chamfer or fillet is drawn on that layer. If they are on different layers, the chamfer or fillet is drawn on the current layer.

You can choose whether to trim the selected edges to the endpoints of the chamfer lines and fillet arcs or not.

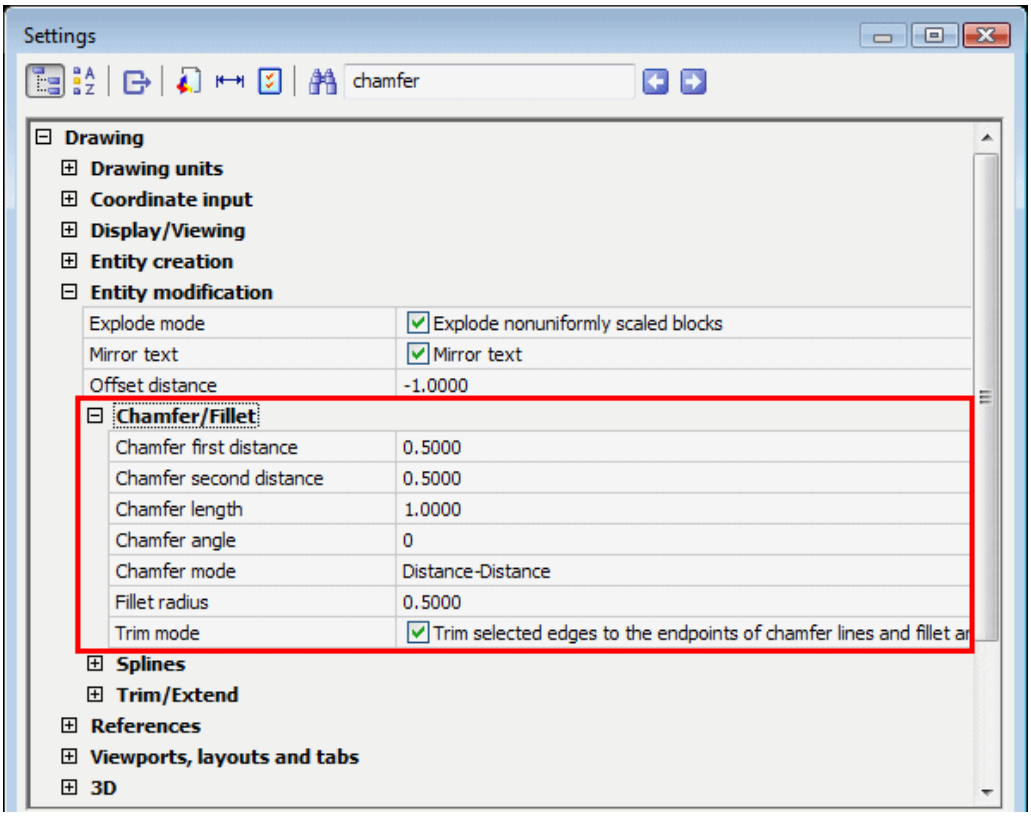


Chamfered rectangle (left) and filleted rectangle (right) with trimmed edges.

Chamfer and fillet settings

1. Launch the Settings Manager.
2. Do one of the following:

- In the *Drawings* settings category, choose *Chamfer/Fillet* in the *Entity Modification* settings sub-category.
- Type *chamfer* or *fillet* in the search field on the *Settings* dialog window.


Settings
chamfer
Drawing
Drawing units
Coordinate input
Display/Viewing
Entity creation
Entity modification
Explode mode
Explode nonuniformly scaled blocks
Mirror text
Mirror text
Offset distance
-1.0000
Chamfer/Fillet
Chamfer first distance
0.5000
Chamfer second distance
0.5000
Chamfer length
1.0000
Chamfer angle
0
Chamfer mode
Distance-Distance
Fillet radius
0.5000
Trim mode
Trim selected edges to the endpoints of chamfer lines and fillet ar
Splines
Trim/Extend
References
Viewports, layouts and tabs
3D

Chamfering Entities

Chamfering Entities

The Chamfer command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In Bricscad you can choose between two chamfer methods:

- *distance-distance*: specify how far to trim the entities back from their intersection
- *distance-angle*: specify the length of the chamfer and the angle it forms along the first entity.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer the entire polyline.

NOTE It is not possible to chamfer segments of different polylines. Use the *Join* option of the Edit Polyline tool to create a single polyline.

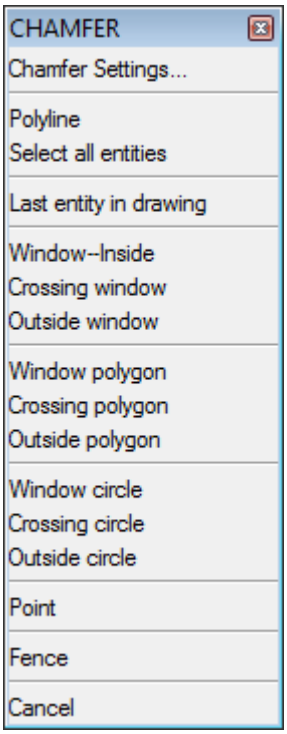
Chamfering using the distance-distance method

1. Do one of the following:

- Click the Chamfer tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Chamfer in the *Modify* menu.
- Type *chamfer* or *CHA* in the command bar, then press Enter.

  The command bar reads: Chamfer (<current chamfer settings>):
  Settings/Polyline/<Select first entity>:

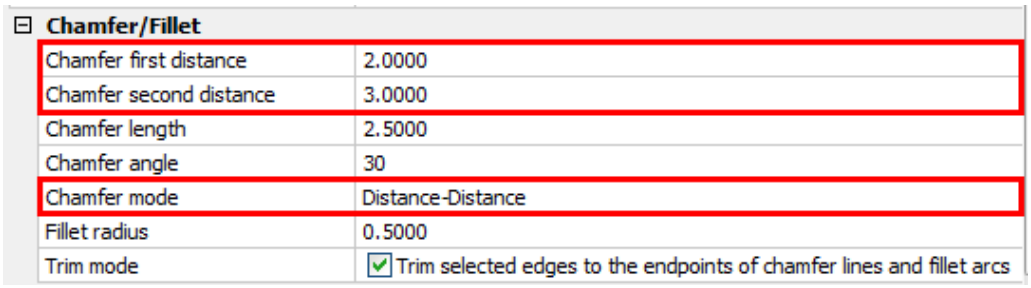
  A prompt box displays:



2. Choose *Chamfer Settings ...* in the prompt box or type *S* and press Enter.

3. In the *Settings* dialog window:

- Specify the *Chamfer first distance*.
- Set the *Chamfer second distance*.
- Set the *Chamfer mode* to *Distance-Distance.*



4. Close the Settings dialog window.

The command bar reads: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

5. Select the first entity or polyline segment.

The command window reads: Select second entity.

6. Select the second entity or polyline segment.

The chamfer is created.

Chamfering using the length-angle method

1. Do one of the following:

- Click the Chamfer tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Chamfer in the *Modify* menu.
- Type *chamfer* or *CHA* in the command bar, then press Enter.

  The command bar reads: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

  A prompt box displays.

2. Choose *Chamfer Settings ...* in the prompt box or type *S* and press Enter.

3. In the *Settings* dialog window:

- Specify the *Chamfer length*.
- Set the *Chamfer angle*.
- Set the *Chamfer mode* to *Length-Angle.*

Chamfer/Fillet	
Chamfer first distance	0.5000
Chamfer second distance	0.5000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Length-Angle
Fillet radius	0.5000
Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

4. Close the Settings dialog window.

The command bar reads: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

5. Select the first entity or polyline segment.

The command window reads: Select second entity.

6. Select the second entity or polyline segment.

The chamfer is created.

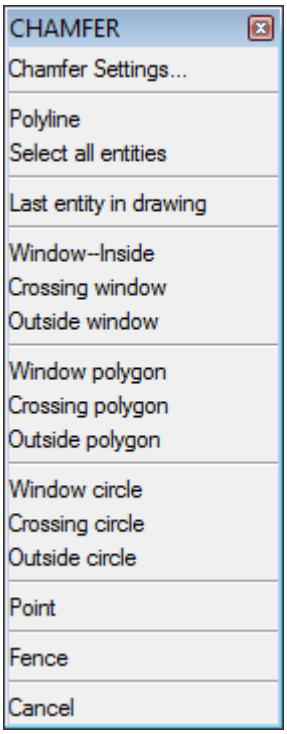
Chamfering all vertices of a polyline

1. Do one of the following:

- Click the Chamfer tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Chamfer in the *Modify* menu.
- Type *chamfer* or *CHA* in the command bar, then press Enter.

  The command bar reads: Chamfer (<current chamfer settings>):
  Settings/Polyline/<Select first entity>:

  A prompt box displays:



2. (option) Adjust the *Chamfer Settings*.

3. Choose *Polyline* in the prompt box or type *P* and press Enter.

The command bar reads: Select 2D polyline to chamfer:

4. Select a polyline.

All vertices of the selected polyline are chamfered.

NOTE When the chamfer method is *distance-angle*, the direction of the polyline defines which is the first entity of a vertex. See drawing rectangles for more information about the direction of closed polylines.

Filleting Entities

Filleting Entities

The Fillet command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

NOTE It is not possible to fillet segments of different polylines. Use the *Join* option of the Edit Polyline tool to create a single polyline.

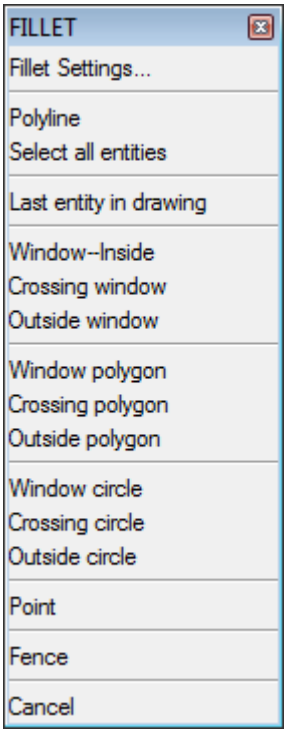
Filleting two entities or polyline segments

1. Do one of the following:

- Click the Fillet tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Fillet in the *Modify* menu.
- Type *fillet* or *F* in the command bar, then press Enter.

  The command bar reads: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>:

  A prompt box displays:



2. Choose *Fillet Settings ...* in the prompt box or type *S* and press Enter.
3. Specify the *Fillet radius* in the *Settings* dialog window:

Chamfer/Fillet	
Chamfer first distance	2.0000
Chamfer second distance	3.0000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Distance-Distance
Fillet radius	2.0000
Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

4. Close the Settings dialog window.

The command bar reads: Chamfer (<current fillet settings>): Settings/Polyline/<Select first entity>:

5. Select the first entity or polyline segment.

The command window reads: Select second entity.

6. Select the second entity or polyline segment.

The fillet is created.

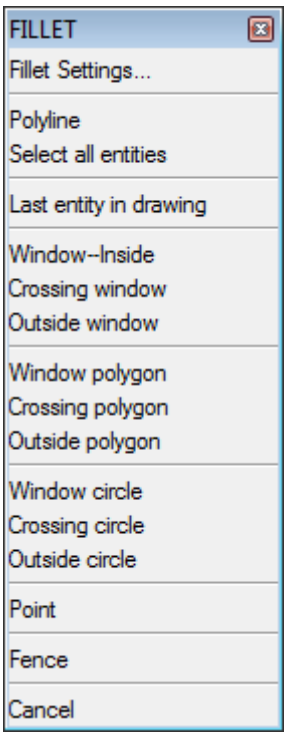
Filleting all vertices of a polyline

1. Do one of the following:

- Click the Fillet tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Fillet in the *Modify* menu.
- Type *fillet* or *F* in the command bar, then press Enter.

The command bar reads: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>:

A prompt box displays:



2. (option) Adjust the *Fillet Settings*.

3. Choose *Polyline* in the prompt box or type *P* and press Enter.

The command bar reads: Select 2D polyline to chamfer:

4. Select a polyline.

All vertices of the selected polyline are filleted.

Filleting two parallel lines

1. Do one of the following:

- Click the Fillet tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose Fillet in the *Modify* menu.
- Type *fillet* or *F* in the command bar, then press Enter.

The command bar reads: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>:

2. Select the first entity (line or ray).

The command window reads: Select second entity:

3. Select the second entity (line, ray or infinite line).

The fillet is executed. The length of the second entity is adjusted.

Editing Polylines

Overview

The Edit Polyline tool can modify any type of two-dimensional or three-dimensional polyline: such as rectangles, polygons and donuts, as well as three-dimensional entities such as pyramids, cylinders and spheres

Editing a polyline can be: opening or closing it, changing its overall width or the widths of individual segments, converting a polyline with straight line segments into a flowing curve or an approximation of a spline.

The Edit Polyline tool lets you edit individual vertices, adding, removing or moving vertices. You can also add new segments to an existing polyline, change the linetypes of a polyline and reverse the direction or order of the vertices.

To modify a polyline, you first select the polyline, and then select a polyline editing option. The available options vary depending on whether the selected polyline is a two-dimensional or three-dimensional entity.

If the selected entity is not a polyline, the Edit Polyline tool provides the option of turning it into one. Only arcs and lines can be converted into polylines. If a series of arcs or lines are connected endpoint to endpoint, they can be merged into one polyline.

Converting an entity into a polyline

Lines and arcs can be converted into single-segment polyline.

Converting an entity into a polyline

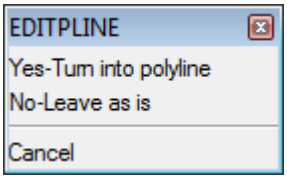
1. Do one of the following:

- Click the Edit Polyline tool button ( ) on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

Select a line or arc.
The command bar reads: The entity selected is not a polyline. Turn it into one? <Y>

A prompt box displays:



2. Do one of the following:

- Press Enter to accept the default option.
- Choose *Yes-Turn into polyline* in the prompt box.

3. Press Enter to conclude the Edit Polyline command.

NOTE The Explode tool turns a single segment polyline back into a line or arc.

Opening and closing polylines

When you close a polyline, a straight polyline segment is drawn from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment.

When you select a polyline for editing, the prompt box displays either the *Open* or *Close* option, depending on whether the polyline you select is closed or open.

The following entities are created as closed polylines:

- rectangles
- polygons
- donuts
- revision clouds

You can open or close a polyline using the Edit Polyline tool or in the Bricscad Properties bar.

To close or open a polyline

1. Do one of the following:

- Click the Edit Polyline tool button () on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

   The command bar reads: Select polyline to edit.

2. Click the polyline you want to close or open.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. To close (or open) a polyline do one of the following:

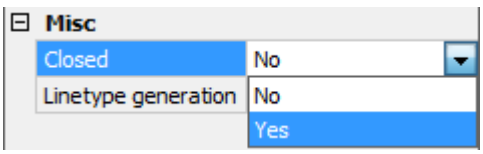
- Type *C* (or *O*) in the command bar, then press Enter.
- Choose *Close* (or *Open*) in the prompt box.

Opening and closing a polyline using the Bricscad Properties bar

1. Select the polyline.

   The current properties of the selected polyline display in the Bricscad Properties bar.

2. Select *Closed* in the *Misc* settings category in the *Bricscad Properties* bar.
3. Click the down arrow and select *Yes* (or *No*).



Joining Polylines

Using the *Join* option of the Edit Polyline tool you can add an arc, a line or a polyline entity to an existing open polyline, forming one continuous polyline entity.

To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline.

When you join an entity to a polyline, the *width* of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

- A line or an arc inherits the width from the polyline segment to which it is joined.
- A polyline joined to a tapered polyline retains its own width values.
- A polyline joined to a uniform-width polyline inherits the width from the polyline to which it is joined.

To join an arc, line, or polyline to an existing polyline

1. Do one of the following:

- Click the Edit Polyline tool button () on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the parent polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. Choose *Join* in the prompt box or type *J* and press Enter.

The command bar reads: Select entities:

4. Select the entity to join, then right click or press Enter.

5. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

NOTE You can select multiple entities in step 3 on condition they form one chain with the parent polyline.

Changing the polyline width

Changing the polyline width

The *Width* option of the Edit Polyline tool applies a uniform width to the entire entity, while the *Taper* option tapers the polyline uniformly alongs its entire length.

To apply a uniform width to an entire polyline

1. Do one of the following:

- Click the Edit Polyline tool button () on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. Choose *Width* in the prompt box or type *W* and press Enter.

The command bar reads: Enter new width for all segments:

4. Type a new width in the command bar and press Enter.

The new width is applied to the entire polyline.

5. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

To taper a polyline uniformly along its length

1. Do one of the following:

- Click the Edit Polyline tool button () on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. Choose *Taper* in the prompt box or type *T* and press Enter.

The command bar reads: Enter new starting polyline width <current starting width>::

4. Type a new starting width in the command bar and press Enter.

The command bar reads: Enter new ending polyline width <current ending width>:

5. Type a new ending width in the command bar and press Enter.

The polyline is tapered uniformly along its length.

6. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Editing polyline vertices

Editing polyline vertices

The *Edit vertices* option of the Edit Polyline tool modifies individual polyline vertices.

When you select this option, the program switches into a special vertex editing mode and places an X on the first vertex. The X indicates the vertex you are editing. The *Next* and *Previous* options move the X to the next or previous vertex. You can edit only one vertex at a time.

When editing vertices, you can modify the polyline in the following ways:

- Convert a straight polyline segment into a curve.
- 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.

Starting the polyline vertex editing mode

1. Do one of the following:

- Click the Edit Polyline tool button () on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

   The command bar reads: Select polyline to edit.

2. Select the polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

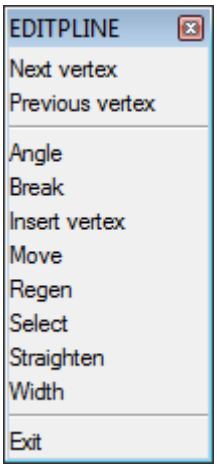
A prompt box displays.

3. Choose *Edit vertices* in the prompt box or type *E* and press Enter.

The command bar reads:

Next/Previous/Angle/Break/Insert/Move/Regen/SElect/Straighten/Width/eXit/<Next>:

The *Editpline* prompt box changes:



A triangle indicates the start point of the polyline. The first vertex of the polyline is selected.

Convert a straight polyline segment into a curve

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment you want to convert.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Choose *Angle* in the prompt box or type *A* and press Enter.
   The prompt box closes.
   The command bar reads: Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <current angle>:
4. Type a new included angle for the segment and press Enter.
   The direction of an arc segment is as follows:

- positive angles: counter clockwise
- negative angle: clockwise direction
- 0° for straight segments

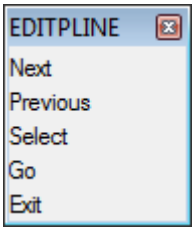
5. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.
6. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Break a polyline into two separate polylines.

1. Start the polyline vertex editing mode.
2. Select the start vertex where you want to break the polyline.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Choose *Break* in the prompt box or type *B* and press Enter.

   The command bar reads: Next/Previous/Select/Go/eXit/<Next>:
   The prompt box changes:



4. (option) Do one of the following to select a second vertex:

- Choose *Next vertex* / *Previous vertex* in the prompt box.
- Choose *Select* in the prompt box or type *S* and press Enter.
  You are prompted to select a vertex.

5. Choose *Go* in the prompt box or type *G* and press Enter.

The polyline s broken into two polylines.

If a second point is selected in step 4 the segment(s) between the selected vertices is (are) deleted.

6. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.

7. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Insert a new vertex in a polyline

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment where you want to insert a vertex.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Choose *Insert vertex* in the prompt box or type *I* and press Enter.
   The command bar reads: Location for new vertex:
4. Specify the location for the new vertex.
   The new vertex is inserted.
5. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.
6. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Move a vertex in a polyline

1. Start the polyline vertex editing mode.
2. Select the vertex you want to move.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Choose *Move* in the prompt box or type *M* and press Enter.
   The command bar reads: New location for vertex.
4. Specify the new location for the selected vertex.
   The vertex is moved.
5. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.

6. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Delete vertices in a polyline

1. Start the polyline vertex editing mode.
2. Select the vertex before the first vertex you want to delete.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Choose *Straighten* in the prompt box or type *S* and press Enter.
   The command bar reads: Straighten: Next/Previous/Select/Go/eXit/<Next>:
4. Do one of the following to select the vertex after the last vertex you want to delete:

- Choose *Next vertex* / *Previous vertex* in the prompt box.
- Choose *Select* in the prompt box or type *S* and press Enter.
  You are prompted to select a vertex.

5. Choose *Go* in the prompt box or type *G* and press Enter.

The vertices between the selected vertices are removed.
A straight segment is drawn between the selected vertices.

6. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.

7. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Change the width of a polyline segment

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment you want to change the width of.
   Choose *Next vertex* / *Previous vertex* in the prompt box to select a vertex.
   The *X* indicates the currently selected vertex.
3. Enter starting width <current width>:
4. Do one of the following:

- Type the new width in the command bar and press Enter.
- Click to define the width graphically.

The command bar reads: Enter ending width <current width>:

5. Do one of the following:

- Type the new width in the command bar and press Enter.
- Click to define the width graphically.

5. Choose *Exit* in the prompt box or type *X* and press Enter to leave the *Polyline vertex editing mode*.

6. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

NOTE The new width is applied when you conclude the Edit Polyline tool in step 6.

Curving and decurving polylines

The *Fit* or *Spline* options of the Edit Polyline tool convert a multi-segment polyline into a smooth curve. The *Fit* option creates a smooth curve connecting all the vertices. The *Spline* option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices.

The *Decurve* option removes *Fit* or *Spline* curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

1. Do one of the following:

- Click the Edit Polyline tool button ( ) on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. Do one of the following:

- Choose *Fit* in the prompt box of type *F* and press Enter.
- Choose *Spline* in the prompt box or type *S* and press Enter.

Original polyline | After applying Fit | After applying Spline

4. Do one of the following:

- Continue editing the selected polyline.
- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

NOTE Use the *Decurve* option in step 3 of the above procedure the restore the original polyline.

Setting the Linetype generation mode

The *Linetype mode* option of the Edit Polyline tool lets you change the way how a dashed linetype is applied to a multi-segment polyline.

To set the Linetype mode

1. Do one of the following:

- Click the Edit Polyline tool button ( ) on the *Modify* toolbar.
- Choose Edit Polyline in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the polyline.

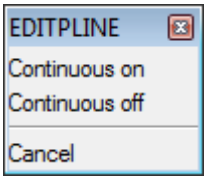
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt box displays.

3. Choose *Linetype* mode in the prompt box or type *L* and press Enter.

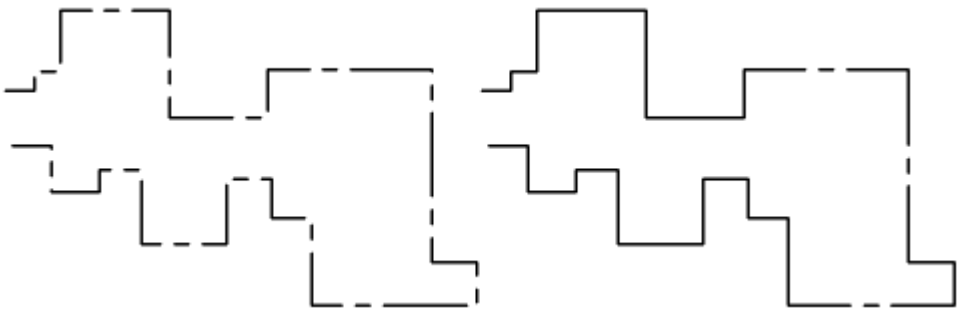
The command bar reads: Linetype continuous along polyline: ON/OFF <current setting>:

The *Editpline* prompt box changes:



4. Do one of the following:

- Choose *Continuous on* or *Continuous off* in the prompt box.
- Type *ON* or *OF* in the command bar, then press Enter.



The same polyline with Continuous On (left) and Continuous Off (right)

5. Do one of the following:

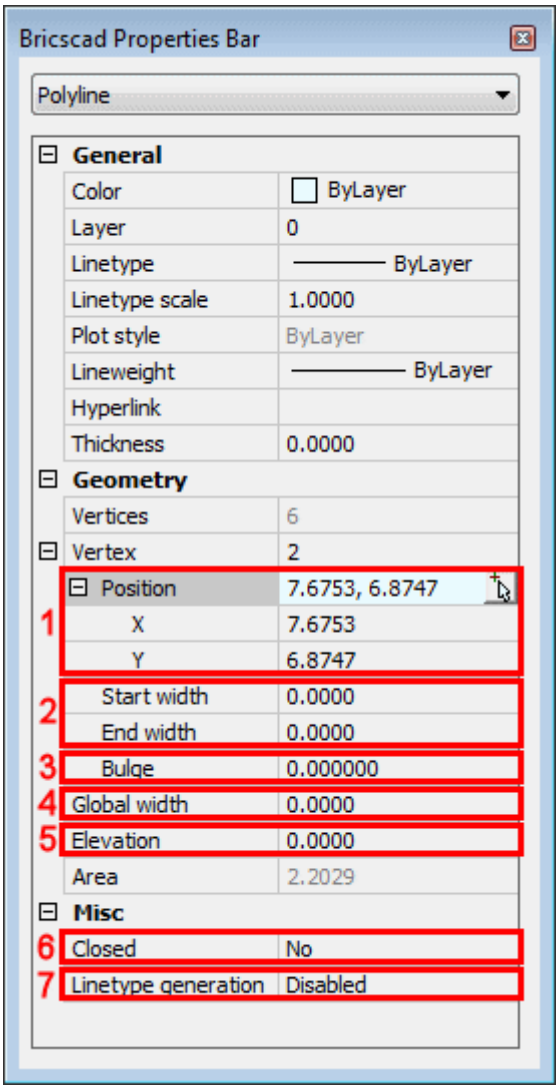
- Continue editing the selected polyline.

- Choose *Done* in the prompt box or press Enter to conclude the Edit Polyline tool.

Editing polylines in the Properties bar

When you select polyline, its current properties display in the Bricscad Properties bar.
In the *Bricscad Properties bar* the following polyline properties can be edited:

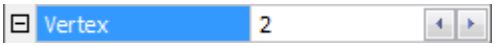
- Move vertices (1)
- Change the width of polyline segments (2)
- Convert straight segments into a curve (3)
- Change the global width (4)
- Change the Elevation (5)
- Open / close the polyline (6)
- Change the Linetype generation mode (7)



To move polyline vertices

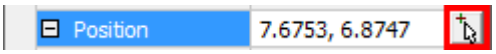
1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the vertex you want to move.

A *X* indicates the currently selected vertex in the drawing.



3. Do one of the following:

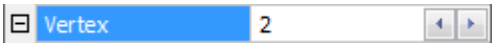
- Adjust the *X-* and/or *Y-field* and press Enter.
- Click the *Vertex Position* field and press the *Position* button to move the vertex in the drawing.



4. Continue editing the polyline or press the Esc key to stop.

To change the width of polyline segments

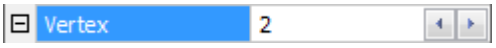
1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.
   A *X* indicates the currently selected vertex in the drawing.



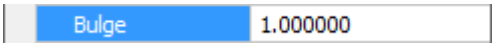
3. Type a new width in the *Start width* field and press Enter.
4. Type a new widith in the *End width* field and press Enter.
5. Continue editing the polyline or press the Esc key to stop.

To convert straight segments into a curve

1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.
   A *X* indicates the currently selected vertex in the drawing.



3. Type a new value in the Bulge field and press Enter.



4. Continue editing the polyline or press the Esc key to stop.

NOTE The bulge is the tangent of one fourth the included angle for an arc segment. A positive bulge creates a counter clockwise arc, while a negative bulge creates an arc with a clockwise direction. Straight segments have a zero bulge.

Converting Entities

Exploding entities

The Explode tool converts complex entities, such as blocks, polylines, solids or dimensions into its component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

In general exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
  Colors and linetypes assigned BYBLOCK may appear different after exploding an entity, because they will adopt the default color and linetype until inserted into another block.

To explode entities

1. Do one of the following:

- Click the Explode tool button () on the *Modify* toolbar.
- Choose Explode in the *Modify* menu.
- Type *explode* in the command bar, then press Enter.

The command window reads: Select entities to explode.

2. Select the entities, then right click or press Enter.

NOTE If you select the entities first, then launch the Explode tool, the selected entities are exploded immediately.

Creating Regions

The Region tool converts closed entities into a solid entity.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts.

Creating regions typically has no visible effect on a drawing. However, if the original entity had a width or lineweight, that information is lost when you create the region.

To create regions

1. Do one of the following:

- Click the Region tool button () in the *Modify* toolbar.
- Choose Region in the *Modify* menu.
- Type *region* in the command bar, then press Enter.

The command bar reads: Select objects:

2. Select the entities, then right click or press Enter.

The command bar displays how many regions have been created.

Flattening Entities

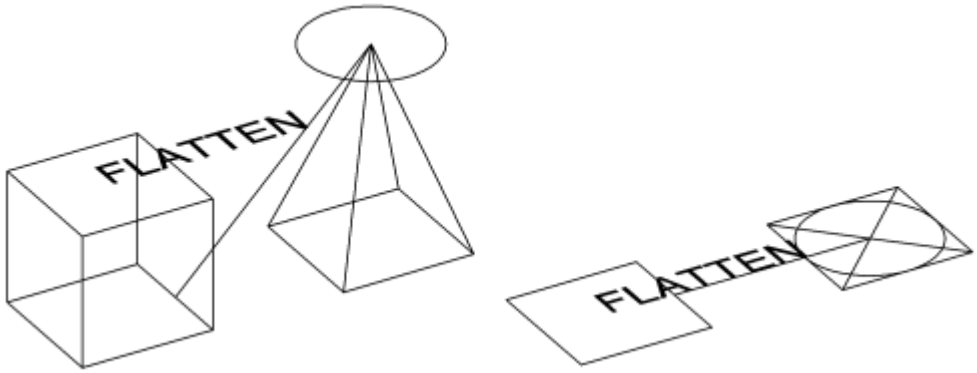
The Flatten tool equals the Z-coordinate of the selected entities, thus flattening three-dimensional entities to two dimensions. All selected entities are flattened onto the current UCS elevation plane, which is not necessarily parallel to the WCS x,y plane.

The following entity types are affected by the Flatten command: line, polyline, arc, circle, ellipse, elliptical arc, point, hatch, block insert, text, Mtext, attribute, dimension, polyface mesh, 3D Face,

Planar entities (such as text, arcs, and two-dimensional polylines) that are not parallel to the current UCS XY-plane are ignored. A message displays in the command bar saying how many entities in the selection set were not parallel to the XY-plane of the UCS.

The Flatten tool can be useful in the following situations:

- to correct the Z-coordinate (elevation)
- to set the Z-coordinate of all entities to zero



Original entities (left) after flattening (right)

To flatten entities

1. Do one of the following:

- Click the Flatten tool button ( ) on the *Modify* toolbar.
- Choose Flatten in the *Modify* menu.
- Type *flatten* in the command bar, then press Enter.

The command bar reads: Select entities to flatten:

2. Select the entities, then right click or press Enter.

The command bar reads: New UCS elevation <current elevation>:

3. Do one of the following:

- Press Enter to accept the current elevation.
- Type a new elevation in the command bar and press Enter.

The command bar displays:

- How many entities were flattened.
- How many entities were not parallel to the XY-plane of the UCS, if any.

Measuring and Dividing Entities

Measuring Entities

The Measure tool places markers - points or blocks - at a specified interval along the length or circumference of an entity. The Measure tool starts placing markers at the closes endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

You measure a circle along its circumference starting from the *Angle Base* value as defined in the *Settings* dialog. If *Angle Base* is set to zero, a circle is measured starting at 3 o'clock (east).

To measure an entity using points

1. Do one of the following:

- Click the Measure tool button () on the *Modify* toolbar.
- Choose Measure in the *Modify* menu.
- Type *measure* in the command bar, then press Enter.

The command bar reads: Select entity to measure:

2. Click the entity you want to measure.

The command bar reads: Block/<Segment length>:

3. To define the measurement interval, do one of the following:

- Type the measurement interval and press Enter.
- Specify two points in the drawing.

Points are placed along the selected entity at the specified interval.

NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To place blocks a specified interval along an entity

1. Do one of the following:

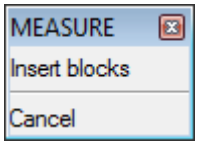
- Click the Measure tool button () on the *Modify* toolbar.
- Choose Measure in the *Modify* menu.
- Type *measure* in the command bar, then press Enter.

The command bar reads: Select entity to measure:

2. Click the entity you want to measure.

The command bar reads: Block/<Segment length>:

A prompt box displays:



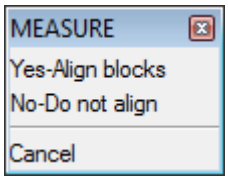
3. Choose *Insert blocks* in the prompt box or type B and press Enter.

The command bar reads: Name of block to insert:

4. Type the name of the block in the command bar and press Enter.

The command bar reads: Align blocks with entity? <Y>:

A prompt box displays:



5. Do one of the following:

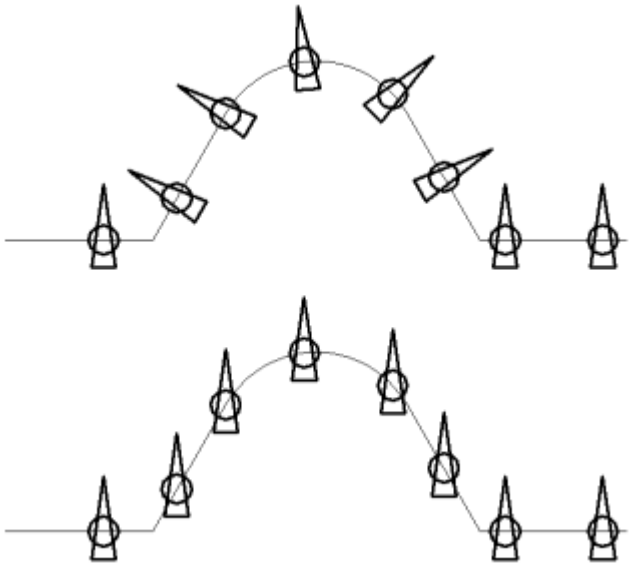
- Choose *Yes-Align blocks* in the prompt box or press Enter to align the blocks with the selected entity.
- Choose *No-Do not align* in the prompt box or type N and press Enter to place the blocks not rotated.

The command bar reads: Segment length:

6. To define the measurement interval, do one of the following:

- Type the measurement interval and press Enter.
- Specify two points in the drawing.

The blocks are placed along the selected entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

Dividing Entities

The Divide tool places markers - points or blocks - along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

To divide an entity using points

1. Do one of the following:

- Click the Divide tool button ( ) on the *Modify* toolbar.
- Choose Divide in the *Modify* menu.
- Type *divide* in the command bar, then press Enter.

The command bar reads: Select entity to divide:

2. Click the entity you want to divide.

The command bar reads: Blocks/<Number of Segments>:

3. Type the number of segments in the command bar and press Enter.

Points are placed along the selected entity to divide the entity.

NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To divide an entity using blocks

1. Do one of the following:

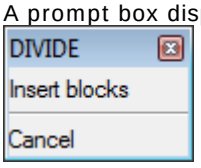
- Click the Divide tool button ( ) on the *Modify* toolbar.
- Choose Divide in the *Modify* menu.
- Type *divide* in the command bar, then press Enter.

The command bar reads: Select entity to divide:

2. Click the entity you want to divide.

The command bar reads: Blocks/<Number of Segments>:

A prompt box displays:



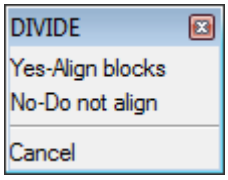
2. Choose *Insert blocks* in the prompt box or type B and press Enter.

The command bar reads: Name of block to insert:

3. Type the name of the block in the command bar and press Enter.

The command bar reads: Align blocks with entity? <Y>:

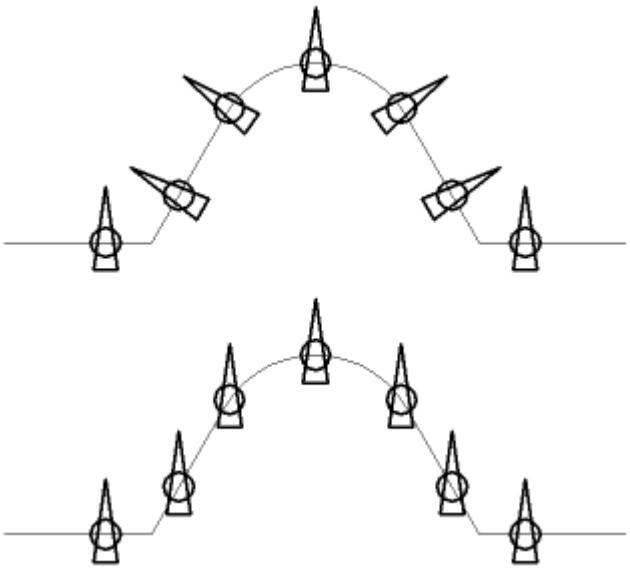
A prompt box displays:



4. Do one of the following:

- Choose *Yes-Align blocks* in the prompt box or press Enter to align the blocks with the selected entity.
- Choose *No-Do not align* in the prompt box or type N and press Enter to place the blocks not rotated.

The blocks are placed along the selected entity to divide the entity.



Blocks aligned (top) or not aligned (bottom)