AutoCAD® 2008

# Command Reference

Autodesk®

January 2007

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## GOVERNMENT USE

# Contents

# Commands

You can start a command by using one of the following methods:

- Click the command name on a menu, toolbar, status bar, or shortcut menu.
- Enter the command name or command alias at the Command prompt and press ENTER or SPACEBAR.

The *acad.pgp* file lists the command aliases. To access the *acad.pgp*, on the Tools menu, click Customize ➤ Edit Program Parameters (*acad.pgp*).

In this *Command Reference*, near the beginning of each command description is a command access section that lists the specific ways you can start that command.

## In this part

- V Commands
- W Commands
- X Commands
- Z Commands

# 3D Commands

**In this chapter**

- 3DSWIVEL
- 3DWALK
- 3DZOOM

# 3D

## Quick Reference

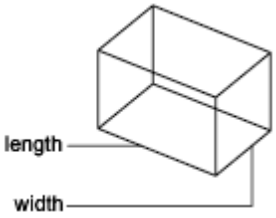
Creates three-dimensional polygon mesh objects in common geometric shapes that can be hidden, shaded, or rendered

**3d**
Enter an option
[Box (page 5)/Cone (page 7)/DIsh (page 9)/DOme (page 9)/Mesh (page 10)/Pyramid (page 11)/Sphere (page 12)/Torus (page 14)/Wedge (page 15)]:

### Box

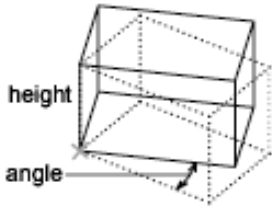
Creates a 3D box polygon mesh.

Specify corner point of box:
Specify length of box: *Specify a distance*
Specify width of box or [Cube]: *Specify a distance or enter* **c**




### Width

Specifies the width of the box. Enter a distance or specify a point relative to the corner point of the box.




Specify height of box: *Specify a distance*

Specify rotation angle of box about the Z axis or [Reference]: *Specify an angle or enter* **r**

**Rotation Angle** Rotates the box about the first corner specified. If you enter **0**, the box remains orthogonal to the current *X* and *Y* axes.

**Reference** Aligns the box with other objects in the drawing or relative to an angle you specify. The base point for the rotation is the first corner of the box.

Specify the reference angle <0>: *Specify a point, enter an angle, or press* ENTER

You can define a reference angle by specifying two points or an angle from the *X* axis on the *XY* plane. For example, you can rotate the box to align two specified points on the box with a point on another object. After defining a reference angle, specify a point for the reference angle to align with. The box then rotates around the first corner relative to the angle of rotation specified for the reference angle.

If you enter **0** as a reference angle, the new angle alone determines the rotation of the box.

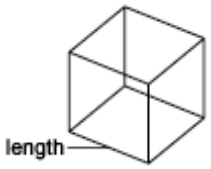
Specify the new angle: *Specify a point or enter an angle*

To specify the new angle of rotation, specify a point relative to the base point. The base point for the rotation is the first corner of the box. The box rotates by the angle between the reference angle and the new angle. If you want to align the box with another object, specify two points on the target object to define the new angle of rotation for the box.

If the reference angle of rotation is 0, the box rotates the angular distance entered relative to the first corner of the box.

### Cube

Creates a cube using the length for the width and height of the box.




Specify rotation angle of box about the Z axis or [Reference]: *Specify an angle or enter* **r**

**Rotation Angle** Rotates the cube about the first corner of the box. If you enter **0**, the box remains orthogonal to the current *X* and *Y* axes.

**Reference** Aligns the box with other objects in the drawing or relative to an angle you specify. The base point for the rotation is the first corner of the box.

Specify the reference angle <0>: *Specify a point, enter an angle, or press* ENTER

You can define a reference angle by specifying two points or an angle from the *X* axis on the *XY* plane. For example, you can rotate the box to align two specified points on the box with a point on another object. After defining a reference angle, specify a point for the reference angle to align with. The box then rotates around the first corner relative to the angle of rotation specified for the reference angle.

If you enter **0** as a reference angle, the new angle alone determines the rotation of the box.

Specify the new angle: *Specify a point or enter an angle*

To specify the new angle of rotation, specify a point relative to the base point. The base point for the rotation is the first corner of the box. The box rotates the angular distance between the reference angle and the new angle. If you want to align the box with another object, specify two points on the target object to define the new angle of rotation for the box.

If the reference angle of rotation is 0, the box rotates the angular distance entered relative to the first corner point of the box.

### Cone

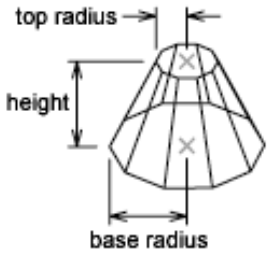
Creates a cone-shaped polygon mesh.


Specify center point for base of cone: *Specify a point (1)*
Specify radius for base of cone or [Diameter]: *Specify a distance or enter* **d**

### Radius for Base

Defines the base of the cone by its radius.

Specify radius for top of cone or [Diameter] <0>: *Specify a distance, enter* **d**, *or press* ENTER




**Radius for Top** Defines the top of the cone by its radius. A value of 0 produces a cone. A value greater than 0 produces a truncated cone.

Specify height of cone: *Specify a distance*

Enter number of segments for surface of cone <16>: *Enter a value greater than 1 or press*ENTER

**Diameter for Top** Defines the top of the cone by its diameter. A value of 0 produces a cone. A value greater than 0 produces a truncated cone.

Specify diameter for top of cone <0>: *Specify a distance or press* ENTER

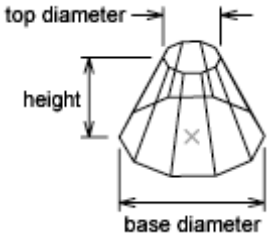
Specify height of cone: *Specify a distance*

Enter number of segments for surface of cone <16>: *Enter a value greater than 1 or press* ENTER

### Diameter for Base

Defines the base of the cone by its diameter.

Specify diameter for base of cone: *Specify a distance*
Specify radius for top of cone or [Diameter] <0>: *Specify a distance, enter* **d**, *or press* ENTER




**Radius for Top** Defines the top of the cone by its radius. A value of 0 produces a cone. A value greater than 0 produces a truncated cone.

Specify height of cone: *Specify a distance*

Enter number of segments for surface of cone <16>: *Enter a value greater than 1 or press*ENTER

**Diameter for Top** Defines the top of the cone by its diameter. A value of 0 produces a cone. A value greater than 0 produces a truncated cone.

Specify diameter for top of cone <0>: *Specify a distance*

Specify height of cone: *Specify a distance*

Enter number of segments for surface of cone <16>: *Enter a value greater than 1 or press*ENTER

### Dish

Creates the lower half of a spherical polygon mesh.

Specify center point of dish: *Specify a point (1)*
Specify radius of dish or [Diameter]: *Specify a distance or enter* **d**
**Radius** Defines the dish by its radius.

Enter number of longitudinal segments for surface of dish <16>: *Enter a value greater than 1 or press* ENTER

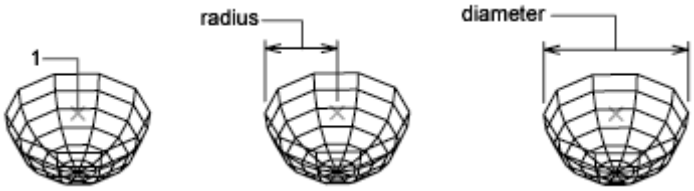
Enter number of latitudinal segments for surface of dish <8>: *Enter a value greater than 1 or press* ENTER

**Diameter** Defines the dish by its diameter.

Specify diameter of dish: *Specify a distance*

Enter number of longitudinal segments for surface of dish <16>: *Enter a value greater than 1 or press*ENTER

Enter number of latitudinal segments for surface of dish <8>: *Enter a value greater than 1 or press*ENTER




### Dome

Creates the upper half of a spherical polygon mesh.

Specify center point of dome: *Specify a point (1)*
Specify radius of dome or [Diameter]: *Specify a distance or enter* **d**
**Radius** Defines the dome by its radius.

Enter number of longitudinal segments for surface of dome: *Enter a value greater than 1 or press* ENTER

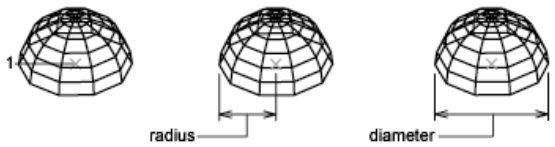
Enter number of latitudinal segments for surface of dome <8>: *Enter a value greater than 1 or press*ENTER

**Diameter** Defines the dome by its diameter.

Specify diameter of dome: *Specify a distance*

Enter number of longitudinal segments for surface of dome <16>: *Enter a value greater than 1 or press*ENTER

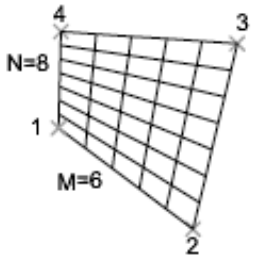
Enter number of latitudinal segments for surface of dome <8>: *Enter a value greater than 1 or press* ENTER




### Mesh

Creates a planar mesh whose *M* and *N* sizes determine the number of lines drawn in each direction along the mesh. The *M* and *N* directions are similar to the *X* and *Y* axes of an *XY* plane.

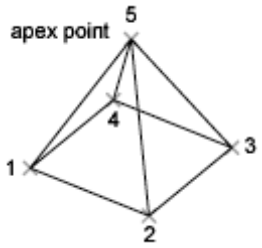
Specify first corner point of mesh: *Specify a point (1)*
Specify second corner point of mesh: *Specify a point (2)*
Specify third corner point of mesh: *Specify a point (3)*
Specify fourth corner point of mesh: *Specify a point (4)*
Enter mesh size in the M direction: *Enter a value between 2 and 256*
Enter mesh size in the N direction: *Enter a value between 2 and 256*



### Pyramid

Creates a pyramid or a tetrahedron.

Specify first corner point for base of pyramid: *Specify a point (1)*
Specify second corner point for base of pyramid: *Specify a point (2)*
Specify third corner point for base of pyramid: *Specify a point (3)*
Specify fourth corner point for base of pyramid or [Tetrahedron]: *Specify a point (4) or enter* **t**




### Fourth Corner Point

Defines the fourth corner point of the base of a pyramid.

Specify apex point of pyramid or [Ridge/Top]: *Specify a point (5) or enter an option*

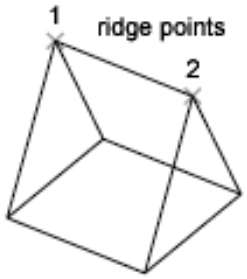
The *Z* value of the point specified determines the height for the pyramid's apex, top, or ridge line.

**Apex Point** Defines the top of the pyramid as a point (apex).

**Ridge** Defines the top of the pyramid as a ridge line. The two endpoints must lie in the same direction as the base points to prevent a self-intersecting wireframe.

Specify first ridge end point of pyramid: *Specify a point (1)*

Specify second ridge end point of pyramid: *Specify a point (2)*




**Top** Defines the top of the pyramid as a rectangle. If the top points cross, they create a self-intersecting polygon mesh.

Specify first corner point for top of pyramid: *Specify a point*

Specify second corner point for top of pyramid: *Specify a point*

Specify third corner point for top of pyramid: *Specify a point*

Specify fourth corner point for top of pyramid: *Specify a point*

### Tetrahedron

Creates a tetrahedral polygon mesh.

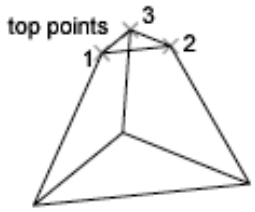
Specify apex point of tetrahedron or [Top]: *Specify a point or enter* **t**
**Apex Point** Defines the top of the tetrahedron as a point (apex).

**Top** Defines the top of the tetrahedron as a triangle. If the top points cross, they create a self-intersecting polygon mesh.

Specify first corner point for top of tetrahedron: *Specify a point (1)*

Specify second corner point for top of tetrahedron: *Specify a point (2)*

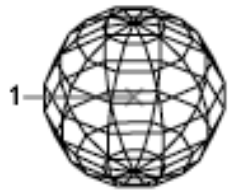
Specify third corner point for top of tetrahedron: *Specify a point (3)*




### Sphere

Creates a spherical polygon mesh.

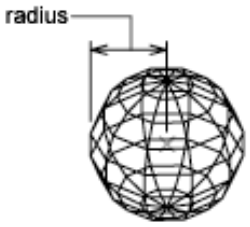
Specify center point of sphere: *Specify a point (1)*
Specify radius of sphere or [Diameter]: *Specify a distance or enter* **d**




**Radius** Defines the sphere by its radius.

Enter number of longitudinal segments for surface of sphere <16>: *Enter a value greater than 1 or press* ENTER

Enter number of latitudinal segments for surface of sphere <16>: *Enter a value greater than 1 or press* ENTER

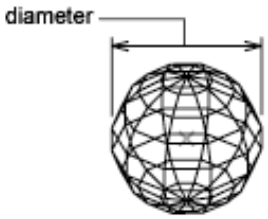



**Diameter** Defines the sphere by its diameter.

Specify diameter of sphere: *Specify a distance*

Enter number of longitudinal segments for surface of sphere <16>: *Enter a value greater than 1 or press* ENTER

Enter number of latitudinal segments for surface of sphere <16>: *Enter a value greater than 1 or press* ENTER



### Torus

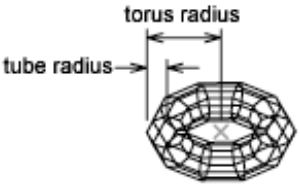
Creates a toroidal polygon mesh that is parallel to the *XY* plane of the current UCS.

Specify center point of torus: *Specify a point (1)*
Specify radius of torus or [Diameter]: *Specify a distance or enter* **d**


The radius of the torus is measured from its center point to its outside edge, not to the center of the tube.

### Radius

Defines the torus by its radius.




Specify radius of tube or [Diameter]: *Specify a distance or enter* **d**

The radius of the tube of the torus is measured from the center of the tube to the outside edge of the tube.

**Radius** Defines the tube by its radius.

Enter number of segments around tube circumference <16>: *Enter a value greater than 1 or press*ENTER

Enter number of segments around torus circumference <16>: *Enter a value greater than 1 or press*ENTER

**Diameter** Defines the tube by its diameter.

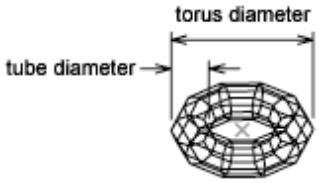
Specify diameter of tube: *Specify a distance*

Enter number of segments around tube circumference <16>: *Enter a value greater than 1 or press*ENTER

Enter number of segments around torus circumference <16>: *Enter a value greater than 1 or press*ENTER

### Diameter

Defines the torus by its diameter.




Specify diameter of torus: *Specify a distance*
Specify radius of tube or [Diameter]: *Specify a distance or enter* **d**

The radius of the tube of the torus is measured from the center of the tube to the outside edge of the tube.

**Radius** Defines the tube by its radius.

Enter number of segments around tube circumference <16>: *Enter a value greater than 1 or press*ENTER

Enter number of segments around torus circumference <16>: *Enter a value greater than 1 or press*ENTER

**Diameter** Defines the tube by its diameter.

Specify diameter of tube: *Specify a distance*

Enter number of segments around tube circumference <16>: *Enter a value greater than 1 or press*ENTER

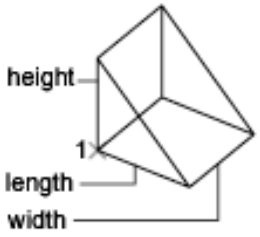
Enter number of segments around torus circumference <16>: *Enter a value greater than 1 or press* ENTER

### Wedge

Creates a right-angle, wedge-shaped polygon mesh with a sloped face tapering along the *X* axis.

Specify corner point of wedge: *Specify a point (1)*
Specify length of wedge: *Specify a distance*
Specify width of wedge: *Specify a distance*
Specify height of wedge: *Specify a distance*
Specify rotation angle of wedge about the Z axis: *Specify an angle*

The base point for the rotation is the corner point of the wedge. If you enter **0**, the wedge remains orthogonal to the current UCS plane.




# 3DALIGN

## Quick Reference

Aligns objects with other objects in 2D and 3D

Modeling 
Modify ➤ 3D Operations ➤ 3D AlignAt the Command prompt, enter 3dalign.

**3dalign**

Select objects: *Select the objects to align and press* ENTER
Specify source plane and orientation . . .

You can specify either one, two or three points for the source object. Then, you can specify either one, two, or three points for the destination. The selected object is moved and rotated so that the base points, and the *X* and *Y* axes of the source and destination align in 3D space. 3DALIGN works with dynamic UCS (DUCS), so you can dynamically drag the selected objects and align them with the face of a solid object.

Specify base point or [Copy]: *Specify a point or enter***c***to create a copy*

The base point of the source object will be moved to the base point of the destination.

Specify second point or [Continue] <C>: *Specify a point on the object's X axis, or press ENTER to skip forward to specifying destination points*

The second point specifies a new *X* axis direction within a plane parallel to the *XY* plane of the current UCS. If you press ENTER instead of specifying a second point, the *X* and *Y* axes are assumed to be parallel with the *X* and *Y* axes of the current UCS.

Specify third point or [Continue] <C>: *Specify a point on the object's positive XY plane, or press ENTER to skip forward to specifying destination points*

The third point fully specifies the orientation of the *X* and *Y* axes of the source object that will be aligned with the destination plane.

Specify destination plane and orientation . . .
Specify first destination point: *Specify a point*

This point defines the destination of the base point of the source object.

Specify second source point or [eXit] <X>: *Specify a point for the X axis of the destination or press* ENTER

The second point specifies a new *X* axis direction for the destination within a plane parallel to the *XY* plane of the current UCS. If you press ENTER instead of specifying a second point, the *X* and *Y* axes of the destination are assumed to be parallel with the *X* and *Y* axes of the current UCS.

Specify third destination point or [eXit] <X>: *Specify a point for the destination's positive XY plane, or press* ENTER

The third point fully specifies the orientation of the *X* and *Y* axes of the destination plane.

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**NOTE** If the destination is a plane on an existing solid object, you can define the destination plane with a single point by turning on dynamic UCS.

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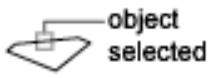
# 3DARRAY

## Quick Reference

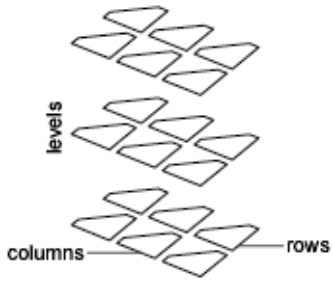
Creates a three-dimensional array

Modify ➤ 3D Operations ➤ 3D ArrayDoes not exist in the menus.
**3darray**
Select objects: *Use an object selection method*




The entire selection set is treated as a single element in the array.

Enter type of array [Rectangular (page 18)/Polar (page 18)] <R>: *Enter an option or press ENTER*




**Rectangular Array** Copies objects in a matrix of rows (*X* axis), columns (*Y* axis), and levels (*Z* axis). An array must have at least two rows or two columns or two levels.

Enter the number of rows (—) <1>: *Enter a positive value or press* ENTER

Enter the number of columns ( | | | ) <1>: *Enter a positive value or press* ENTER

Enter the number of levels ( . . . ) <1>: *Enter a positive value or press* ENTER

Specifying one row requires that more than one column be specified, and vice versa. Specifying one level creates a two-dimensional array.

If you specify more than one row, the following prompt is displayed:

Specify the distance between rows (—): *Specify a distance*

If you specify more than one column, the following prompt is displayed:

Specify the distance between columns ( | | | ): *Specify a distance*

If you specify more than one level, the following prompt is displayed:

Specify the distance between levels ( . . . ): *Specify a distance*

Positive values generate the array along the positive *X*, *Y*, and *Z* axes. Negative values generate the array along the negative *X*, *Y*, and *Z* axes.

**Polar Array** Copies objects about an axis of rotation.

Enter the number of items in the array: *Enter a positive value*

Specify the angle to fill (+=ccw, -=cw) <360>: *Specify an angle or press* ENTER

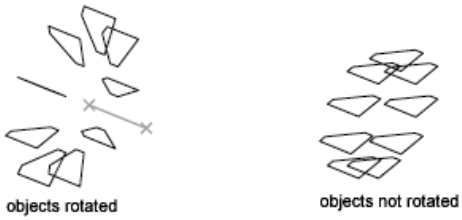
The specified angle determines how far the objects are arrayed about the axis of rotation. A positive number produces a counterclockwise array rotation. A negative number produces a clockwise array rotation.

Rotate arrayed objects? [Yes/No] <Y>: *Enter* **y** *or* **n**, *or press* ENTER

Entering **y** or pressing ENTER rotates each array element.

Specify center point of array: *Specify a point (1)*

Specify second point on axis of rotation: *Specify a point (2)*




# 3DCLIP

## Quick Reference

Starts an interactive 3D view and opens the Adjust Clipping Planes window

**3dclip**

The Adjust Clipping Planes window (page 19) is displayed.

# Adjust Clipping Planes Window

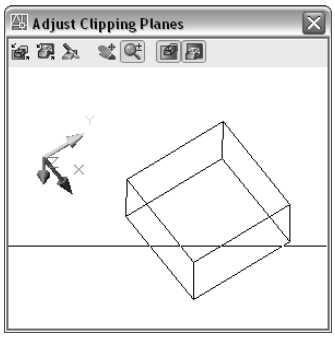
## Quick Reference

**3dclip**

Sets clipping planes for the display in 3D Orbit view. Use the Adjust Clipping Planes toolbar, or right-click in the Adjust Clipping Planes window and select an option from the shortcut menu (page 20).

Click the Close button in the upper-right corner of the window to see the effect of the clipping planes. As you rotate the view, different portions of the objects are clipped as they pass in and out of the clipping planes.

To turn clipping planes off, right-click in the drawing area or the Adjust Clipping Planes window, and check or clear Front Clipping On and Back Clipping On.




**Adjust Clipping Planes Shortcut Menu**

To choose any of the following options, right-click in the Adjust Clipping Planes window.

**Adjust Front Clipping**  Adjusts only the front clipping plane. The line near the bottom of the window adjusts the front clipping plane. If a check mark is displayed next to Front Clipping On, you can see the clipping in the 3D Orbit view as you move the line up or down.

**Adjust Back Clipping**  Adjusts only the back clipping plane. The line near the top of the window adjusts the back clipping plane. If a check mark is displayed next to Back Clipping On, you can see the clipping in the 3D Orbit view as you move the line up or down.

**Create Slice**  Causes the back and front clipping planes to move together, which shows you a "slice" of the objects in the 3D Orbit view. Adjust the front and back clipping planes, and then choose Create Slice. You can then move the front and back clipping planes as one unit.

**Pan**  Displays the pan cursor, a hand cursor that you can use to pan the clipping plane. Hold down the pick button and drag the cursor in any direction. The pan cursor stays active until you click another button.

**Zoom**  Displays the zoom cursor, a magnifying-glass cursor that you can use to enlarge or reduce the clipping plane. To enlarge the image, hold down the pick button and drag the cursor toward the top of Adjust Clipping Planes window. To reduce the image, hold down the pick button and drag the cursor toward the bottom of the window.

**Front Clipping On**  Turns the front clipping plane on or off. A check mark indicates that the front clipping plane is on, and you can see the results of moving the line that adjusts the front clipping plane. Choose this option again to turn the front clipping plane off and remove the check mark.

**Back Clipping On**  Turns the back clipping plane on or off. A check mark indicates that the back clipping plane is on, and you can see the results of moving the line that adjusts the back clipping plane. Choose this option again to turn the back clipping plane off and remove the check mark.

**Reset** Resets the the window and clipping plane to the extents of the drawing.

# 3DCONFIG

## Quick Reference

Provides 3D graphics system configuration settings

**3dconfig**

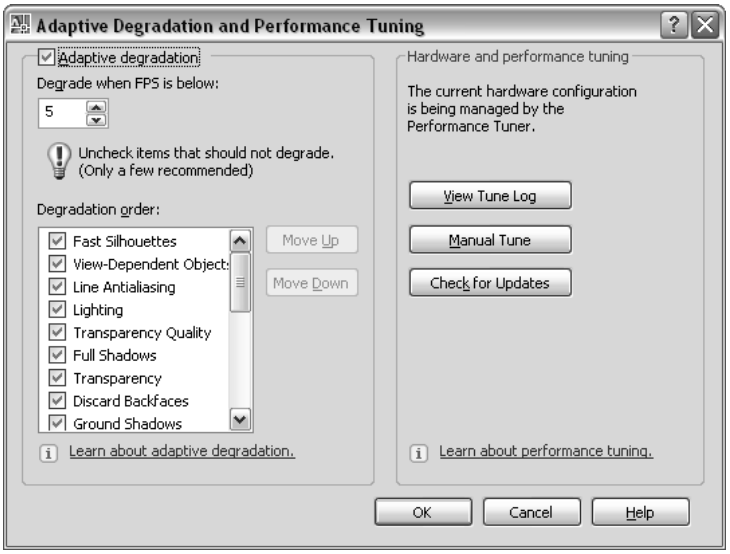
The Adaptive Degradation and Performance Tuning dialog box (page 22) is displayed.

If you enter **-3dconfig** at the command prompt, options are displayed at the command prompt (page 28).

# Adaptive Degradation and Performance Tuning Dialog Box

## Quick Reference

Controls 3D display performance. (This dialog box can also be accessed from the Performance Settings button on the System tab of the Options dialog box.)




Turns adaptive degradation on and off. With adaptive degradation on, if performance goes below the level you specify, effects are turned off or turned down in a certain order until performance returns to an acceptable level.

**Degrade When FPS Is Below** Sets the level at which adaptive degradation begins in frames per second (FPS). The default speed is 5 FPS. Enter a number or choose a number from the list.

**Degradation Chain Order** Specifies the order in which effects are degraded. Items at the top are degraded first. Items that are not checked are not degraded. Click Move Up or Move Down to move a selected item up or down in the chain.

**Move Up** Moves the selected item up in the chain.

**Move Down** Moves the selected item down in the chain.

#### Hardware and Performance Tuning

Specifies hardware settings and performance tuning. The performance tuner reads your system and decides whether to use software or hardware implementation for features that support both. Features that work with your system are turned on, and features that cannot work with your system are turned off. A log file displays the results. You can make manual changes.

**View Tune Log** Displays the Performance Tuner log (page 23).

**Manual Tune** Displays the Manual Performance Tuning dialog box (page 24).

**Check for Updates** Displays a Web page that describes the certification program for graphics cards and display drivers. You can download the latest list of certified cards and drivers at any time.

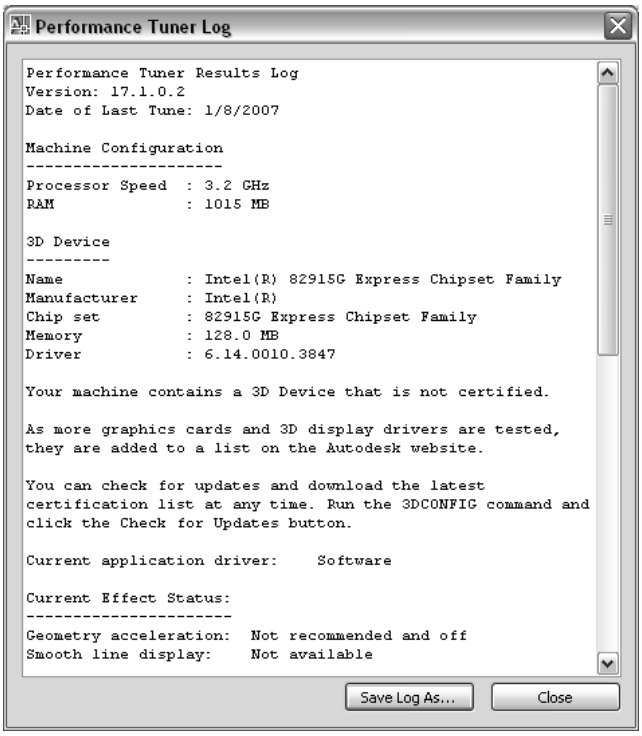
Graphics hardware certification indicates which graphic card and driver combinations that Autodesk has tested to ensure that it supports real-time 3D shading, shadows, smooth-line display, and texture compression features of AutoCAD 2007 (and later) based products. Autodesk certified graphics hardware is better suited for the 3D display features of AutoCAD 2007 and later releases, and non-certified graphics hardware may not support these features or may cause problems during use.

# Performance Tuner Log

## Quick Reference

Tools ➤ Options (Options dialog box, System tab)
**options**

Lists the features that have been disabled, if any. Information includes the system configuration, 3D graphics device, and driver. If your 3D device is certified, Autodesk has tested it with the performance tuner.




**Save Log As**

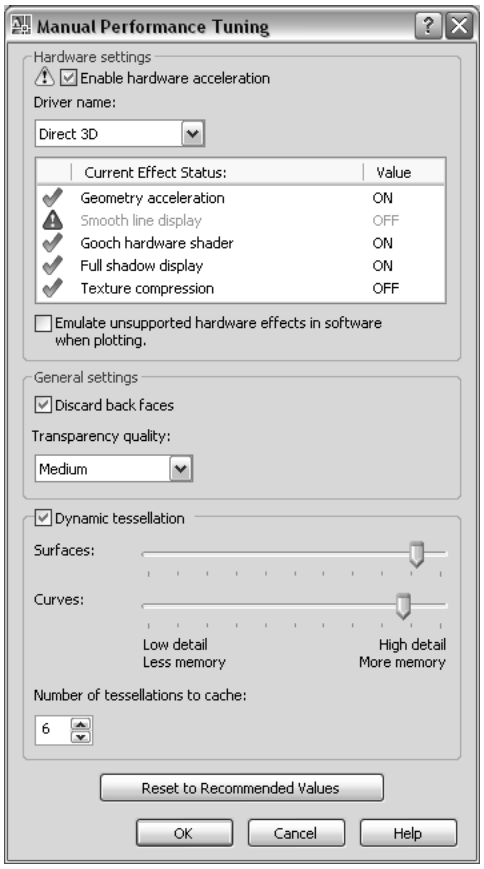
Displays a standard file selection dialog box (page 931).

# Manual Performance Tuning Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, System tab)
**options**

Sets options that affect 3D display performance. These options are set automatically by the performance tuner. You can set them manually in this dialog box.




Sets a driver for the graphics card and sets options for hardware acceleration.

**Enable Hardware Acceleration** Uses hardware acceleration. When this check box is cleared, hardware acceleration is not used, and all the items in this dialog box are unavailable. For information on why hardware acceleration might not be available, see Hardware Acceleration Troubleshooting in the *Driver and Peripheral Guide*.

- Green check mark indicates a graphics card that is certified and can run all features.
- Yellow warning indicates a graphics card that is certified and cannot run all features. If the graphics card is unknown, the performance tuner displays the yellow warning, and you can use hardware acceleration at your own risk.

- Red alert indicates a graphics card that is not certified. The check box is cleared, and Enable Hardware Acceleration is unavailable.

**Driver Name** Sets a driver for the graphics card. (OpenGL, Direct 3D, or other custom drivers that might be installed). If no drivers are certified for the graphics card, "no certified drivers" is displayed. Changes to the driver may affect the Hardware Effects List.

**Hardware Effects List** Lists features and indicates their status and whether they are on or off. A check mark indicates a feature that is supported and on; you can turn it off. A yellow warning indicates that the feature is not recommended; you can turn it on or off. A red alert indicates that the feature is not supported and cannot be turned on.

**NOTE** To display full shadows in the viewport, hardware acceleration is required. When Geometry Acceleration is off, full shadows cannot be displayed in the viewport.

**NOTE** When Texture Compression is turned on, the amount of video memory required to open a drawing that contains materials with images or has attached images is decreased. By using the effect, it can reduce the amount of video memory necessary to display the drawing; the downside to this effect is it may increase the time it takes to load the images the first time that they are accessed and there is a reduction in the quality of the images when they are displayed in the viewport or plotted.

**Emulate Unsupported Hardware Wffects in Software When Plotting** Controls if software emulation is used for unsupported hardware effects when plotting shaded viewports with shadows and other effects that are not supported by your graphics card.

**WARNING** It is recommended that you not use any type of remote access application, such as NetMeeting, Remote Desktop, or VMWare in conjunction with hardware acceleration. Most remote access applications do not support hardware acceleration, and as a result, can cause general display failure and instability. Prior to using a remote access application with AutoCAD, check with the software vendor to see if it supports hardware acceleration and if not, disable hardware acceleration prior to starting the remote access software or use the /NOHARDWARE command line switch to start AutoCAD in Software mode.

### General Settings

Sets performance-related options that are not hardware dependent.

**Discard Back Faces** When checked, the back faces of objects are not drawn.

**Transparency Quality** Adjusts the transparency quality. At the Low setting, a screen-door effect achieves transparency without sacrificing speed. At the Medium setting, the default in software mode, blending improves image quality. The High setting, the default in hardware mode, produces an image free of visual artifacts at the cost of drawing speed.

### Drawing Settings

Sets performance values on a drawing level.

**Dynamic Tessellation** Sets the options that determine the smoothness of the objects in a drawing. Objects are drawn using many short lines (or triangles when drawing spheres). These lines are called *tessellation lines*. Objects in your drawing appear smoother when you use more tessellation lines.

**Surface Tessellation** Determines the amount of detail for surfaces in your drawing. The low end of the slider provides fewer tessellation lines and uses less memory.

**Curve Tessellation** Determines the amount of detail for curves in your drawing. The low end of the slider provides fewer tessellation lines and uses less memory.

**Number of Tessellations to Cache** Configures your system according to memory and performance requirements. The 3D cache always stores at least one tessellation. When this option is set to 1, the tessellation for all viewports is the same; some objects in the drawing may be regenerated as you zoom in and out. Setting this option to 2 or more is useful when you have more than one viewport with different views. Increasing the number requires more memory.

### Reset to Recommended Values

Resets the values to the defaults based on the performance tuner's evaluation of the graphics card.

# -3DCONFIG

## Quick Reference

If you enter **-3dconfig** at the command prompt, the following 3DCONFIG command prompts are displayed.

Enter option: [Adaptive degradation (page 28)/Dynamic tessellation (page 28)/General options (page 29)/acceLeration (page 29)/Plot emulation (page 31)/eXit (page 31)] <Adaptive degradation>:

The prompt is redisplayed until you enter **x** (Exit) or press ESC.

### Adaptive Degradation

Turns off or minimizes display effects when performance drops below the minimum you specify.

Enter mode [ON/OFF] <ON>:

If Adaptive Degradation is on, the following prompt is displayed.

Enter speed to maintain (1-60 fps) <5>:

The main prompt is redisplayed.

### Dynamic Tessellation

Sets the options that determine the smoothness of the objects in a drawing. Objects are drawn using many short lines (or triangles when drawing spheres). These lines are called *tessellation lines*. Objects in your drawing appear smoother when you use more tessellation lines.

Enter mode [ON/OFF] <ON>:

If dynamic tessellation is on, the following prompt is displayed.

Enter option [Surface tessellation/Curve tessellation/Tessellations to cache/eXit] <Surface tessellation>:

The prompt is redisplayed.

**Surface Tessellation** Enter tolerance <92>: *Enter a value between 0 and 100*

Determines the amount of detail for surfaces in your drawing. A higher setting provides more detail but uses more tessellation lines and more memory.

The Dynamic Tessellation prompt is redisplayed.

**Curve Tessellation** Enter tolerance <87>: *Enter a value between 0 and 100*

Determines the amount of detail for curves in your drawing. A higher setting provides more detail but uses more tessellation lines and more memory.

The Dynamic Tessellation prompt is redisplayed.

**Number of Tessellations to Cache** Enter number of tessellations to cache <4>: *Enter a value between 1 and 10*

Configures your system according to memory and performance requirements. The 3D cache always stores at least one tessellation. When this option is set to 1, the tessellation for all viewports is the same; some objects in the drawing may be regenerated as you zoom in and out. Setting this option to 2 or more is useful when you have more than one viewport with different views. Increasing the number requires more memory.

The Dynamic Tessellation prompt is redisplayed.

### General Options

Sets performance-related options that are not hardware dependent.

Configure: General options
Enter option [Discard backfaces/Transparency quality/eXit] <Discard backfaces>:
**Discard Back Faces** Enter mode [ON/OFF] <ON>: *Enter* **on** *to discard back faces*

Discards back faces when drawing objects. You cannot see the effect of discarding back faces on some objects, such as spheres, because you cannot see the back face even when it is present. The effect of discarding back faces is visible on objects such as those that don't have a top. Discarding back faces enhances performance.

The General Options prompt is redisplayed.

**Transparency Quality** Adjusts the transparency quality. At the Low setting, a screen-door effect achieves transparency without sacrificing speed. At the Medium setting, the default in software mode, blending improves image quality. The High setting, the default in hardware mode, produces an image free of visual artifacts at the cost of drawing speed. Materials must also be turned on for transparency to be visible.

The General Options prompt is redisplayed.

### Acceleration

Specifies whether to use software or hardware acceleration in 3D.

Enter option [Hardware/Software/eXit] <Hardware>:

Enter mode [ON/OFF] <ON>: *Specify whether you want anti-alias lines turned on or off*

If you select Hardware, you can also specify whether geometry acceleration and anti-alias lines are turned on or off.

**Hardware** Enter option [Driver name/Geometry acceleration/Antialias lines/Shadows enabled/Texture compression/eXit] <Driver name>:

Specifies hardware acceleration. The hardware graphics card performs most of the drawing tasks in 3D to increase performance.

Enter driver name <wopengl9.hdi>:

The available drivers are *wopengl9.hdi* and *direct3d9.hdi*.

When you select Hardware, the default driver is set to the *wopengl9.hdi* driver that is included with the program.

If you select the Driver Name option, you can select a driver from a list of available hardware-accelerated drivers found in the *Drv* directory. If you want to use a hardware driver from another vendor, it must be supported by the Heidi® Graphics System.

For information on why hardware acceleration might not be available, see Hardware Acceleration Troubleshooting in the *Driver and Peripheral Guide*.

**Software** Specifies software acceleration. The software graphics system performs all of the drawing tasks in 3D.

**Geometry Acceleration** Enter mode [ON/OFF] <ON>:

Specifies whether to use geometry acceleration. Geometry acceleration must be supported by your graphics card. See your vendor documentation for information about your graphics card.

Using geometry acceleration makes more efficient use of the graphics card. With this option turned on, you are drawing in single precision. Use this option only if you know that your drawing is within the single-precision limit.

**Anti-alias Lines** Enter mode [ON/OFF] <ON>:

Specifies using anti-aliasing lines. Lines appear smoother and less gradient.

**Shadows Enabled** Enter mode [ON/OFF] <ON>:

Enables the display of shadows.

---

**NOTE** When Geometry Acceleration is turned off, full shadows cannot be displayed in the viewport.

---

**Texture Compression** Enter mode [ON/OFF] <ON>:

Enables the use of texture compression for images.

**NOTE** When Texture Compression is turned on, the amount of video memory required to open a drawing that contains materials with images or has attached images is decreased. By using the effect, it can reduce the amount of video memory necessary to display the drawing; the downside to this effect is it may increase the time it takes to load the images the first time that they are accessed and there is a reduction in the quality of the images when they are displayed in the viewport or plotted.

#### Plot Emulation

Turns on or off the use of software emulation for unsupported hardware effects during the plotting of a drawing with shaded viewports. The use of software emulation happens when it is turned on, and hardware acceleration is disabled or is enabled, but does not support Shadows.

Enter mode [ON/OFF] <ON>:

The main prompt is redisplayed.

#### Exit

Exits the command.

# 3DCORBIT

## Quick Reference

Starts an interactive 3D view and sets the objects into continuous motion

3D Navigation 
View ➤ Orbit ➤ Continuous OrbitDoes not appear in the menus.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigation Modes ➤ Continuous Orbit (3).
**3dcorbit**
3D Navigate panel, Constrained Orbit flyout

You can view your entire drawing or select one or more objects before starting the command.

Click in a drawing area and drag the pointing device in any direction to start the objects moving in the direction that you're dragging. Release the button

on the pointing device and the objects continue their orbit in the direction that you specified. The speed set for the cursor movement determines the speed at which the objects spin.

You can change the direction of the continuous orbit by clicking and dragging again. You can also change the display of the continuous orbit by right-clicking in the drawing area and choosing an option from the shortcut menu. For example, you can choose Visual Aids ➤ Grid to add a grid to the view without exiting Continuous Orbit.

# 3DDISTANCE

## Quick Reference

Starts the interactive 3D view and makes objects appear closer or farther away

3D Navigation 
View ➤ Camera ➤ Adjust DistanceDoes not appear in the menus.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigation Modes ➤ Adjust Distance (4).
**3ddistance**

3DDISTANCE changes the cursor to a line with one arrow pointing up and one pointing down. Click and drag the cursor vertically toward the top of the screen to move the camera closer to the objects, making them appear larger. Click and drag the cursor vertically toward the bottom of the screen to move the camera away from the objects, making them appear smaller.

# 3DDWF

## Quick Reference

Creates a 3D DWF file of your three-dimensional model and displays it in the DWF Viewer

Standard 

**3ddwf**

The Export 3D DWF dialog box (a standard file selection dialog box (page 931)) is displayed. After you enter a file name and click Save, the Autodesk DWF Viewer is launched and your DWF file is displayed.

You can also change publishing settings from the 3D DWF Publish dialog box (page 33).

**NOTE** The File Type control on the Export 3D DWF dialog box only lists *3D DWF (*.dwf)* when you use the 3DDWF or 3DDWFPUBLISH commands.

# 3D DWF Publish Dialog Box

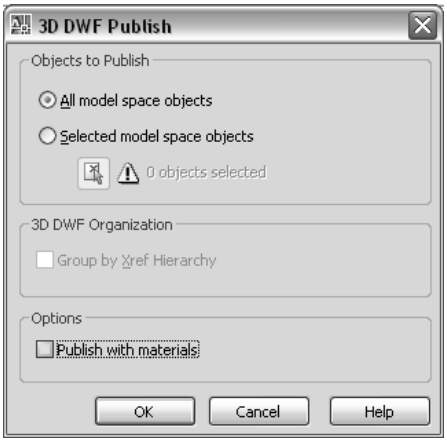
## Quick Reference

Standard 

Issue the 3DDWF command and click Tools ➤ Options from the Export 3D DWF dialog box.

**3ddwf**

Allows you to publish a three-dimensional model as a Design Web Format™ (DWF™) file. 3D DWF Publishing, a technology preview in AutoCAD® 2006, is now a standard AutoCAD feature.



### Objects to Publish

Specifies whether all model space objects or selected model space objects are published to the DWF file.

**All Model Space Objects** All model space objects are published to the 3D DWF file.

**Select Model Space Objects** Allows you to create a selection set of model space objects, including external references, which are published to the 3D DWF file after you click OK.

**Select Objects** Active only when Select Model Space Objects is enabled. Clicking this button minimizes the dialog boxes and lets you select objects in the drawing workspace. When object selection is completed, press ENTER and the 3D DWF Publish dialog box reappears reporting the number of objects selected for publishing.

### 3D DWF Organization

Specifies how to group objects in the DWF file for viewing in Autodesk® DWF™ Viewer.

**Group by Xref Hierarchy** Arrange objects by xref hierarchy in the viewer. If unchecked, externally referenced files are listed in the viewer like any other objects.

### Options

These options let you specify if materials are published in the DWF file.

**Publish with Materials** Most materials that you have assigned to your model will not be published to the 3D DWF file. Materials with procedural maps and some material properties will not export to the 3D DWF file. If your material contains texture mapping, only the Diffuse Map gets published along with its scale and orientation data. Including materials will not significantly affect the size of your 3D DWF files. Default=active.

---

**NOTE** Materials that are not supported in 3D DWF include Bump, Opacity, and Reflection mapping, as well as all forms of procedural materials, which include Checker, Marble, Noise, Speckle, Tile, Waves and Wood.

---

# 3DDWFPUBLISH

## Quick Reference

Obsolete

**3ddwfpublish**

The Export 3D DWF dialog box (a standard file selection dialog box (page 931)) is displayed.

Refer to the *3DDWF* command.

# 3DFACE

## Quick Reference

Creates a three-sided or four-sided surface anywhere in 3D space

Draw ➤ Modeling ➤ Meshes ➤ 3D FaceAt the Command prompt, enter 3dface.

**3dface**

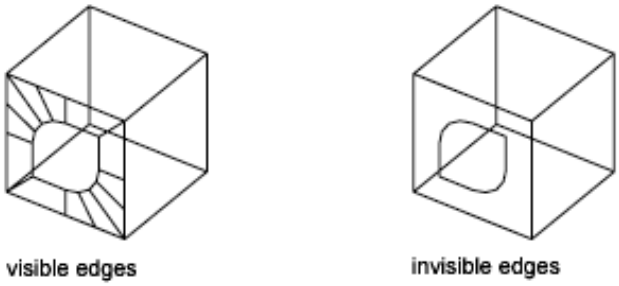
Specify first point (page 35) or [Invisible (page 35)]: *Specify a point (1) or enter* **i**

**First Point** Defines the start point for the 3D surface. After entering the first point, enter the remaining points in a natural clockwise or counterclockwise order to create a normal 3D face. If you locate all four points on the same plane, a planar face is created that is similar to a region object. When you shade or render the object, planar faces are filled.

**Invisible** Controls which edges of a 3D face are visible, allowing for accurate modeling of objects with holes. Entering **i** or **invisible** before the first point of an edge makes the edge invisible.

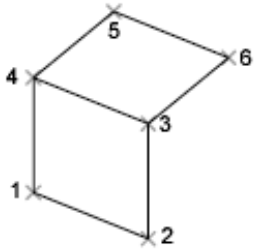
The invisible specification must precede any object snap modes, *XYZ* filters, or coordinate input for that edge. You can create a 3D face in which all edges are invisible. Such a face is a phantom; it does not appear in wireframe presentations but can hide material in line drawings. 3D faces do appear in shaded renderings.

You can combine 3D faces to model complex 3D surfaces.




Specify second point or [Invisible]: *Specify a point (2) or enter* **i**
Specify third point or [Invisible] <exit>: *Specify a point (3), enter* **i,** *or press*ENTER
Specify fourth point or [Invisible] <create three-sided face>: *Specify a point (4), enter* **i**, *or press* ENTER

The Third Point and Fourth Point prompts are repeated until you press ENTER. Specify points 5 and 6 at these repeating prompts. When you finish entering points, press ENTER.




# 3DFLY

## Quick Reference

Interactively changes your view of a 3D drawings so that you appear to be flying through the model

3D Navigation 
View ➤ Walk and Fly ➤ Fly
Start any 3D navigation command, right-click in the drawing area, and click Other Navigations Modes ➤ Fly (7).

**3dfly**
3D Navigate panel, Walk flyout

3DFLY activates a fly mode in the current viewport. You can leave the XY plane as though you're flying through or around the model. On the keyboard, use the four arrow keys, the W (forward), A (left), S (back), and D (right) keys, and the mouse to determine the direction of your fly. For more information, see Walk and Fly Navigation Mappings Dialog Box (page 56).

By default, the Position Locator window (page 54) opens and displays your position in the drawing from a top view.

To control walk and fly settings, use the Walk and Fly Settings dialog box. (page 1546)To specify settings for animating a 3D walk or fly, see Animation Settings Dialog Box (page 54).

# 3DFORBIT

## Quick Reference

Controls the interactive viewing of objects in 3D, using an unconstrained orbit

3D Navigate 
View ➤ Orbit ➤ Free OrbitAt the Command prompt, enter 3DFOrbit.
Press SHIFT+CTRL and click the mouse wheel to temporarily enter 3DFORBIT mode.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigations Modes  Free Orbit (2).
**3dforbit**

3DFORBIT activates a 3D Free Orbit view in the current viewport. If the user coordinate system (UCS) icon is on, a shaded 3D UCS icon representing the current UCS appears in the 3D Orbit view. You can view your entire drawing or select one or more objects before starting the command.

The 3D Free Orbit view displays an arcball, which is a circle divided into four quadrants by smaller circles. When the Enable Orbit Auto Target option is deselected in the shortcut menu, the target of the view stays stationary. The camera location, or point of view, moves around the target. The center of the arcball, not the center of the objects you're viewing, is the target point. Unlike

*3DORBIT*, 3DFORBIT does not constrain the change in view along the *XY* axis or *Z* direction.

**NOTE** You cannot edit objects while the 3DFORBIT command is active.

Moving your cursor over different parts of the arcball changes the cursor icon, indicating the direction in which the view rotates. See 3D Free Orbit Cursor Icons (page 38).

While the command is active, you can access additional 3DORBIT options from a shortcut menu by right-clicking in the drawing area, or choosing buttons on the 3D Navigation toolbar. See 3DOrbit Shortcut Menu (page 43).

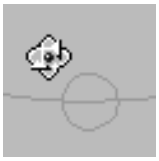
**TIP** While in 3D Orbit mode, you can temporarily enter 3D Free Orbit mode by pressing and holding the SHIFT key.

# 3D Free Orbit Cursor Icons

## Quick Reference

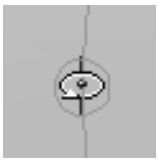
View rotation is determined by the placement and appearance of the cursor as follows:

**Sphere Encircled by Two Lines** When you move the cursor inside the arcball, it changes to a small sphere encircled by two lines. If you click and drag in the drawing area you can move freely around the objects. It works as if your cursor is grabbing a sphere surrounding the objects and dragging it around the target point. You can drag horizontally, vertically, and diagonally using this method.

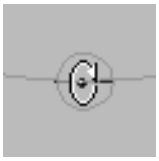


**Circular Arrow** When you move the cursor outside the arcball, it becomes a circular arrow. Clicking outside the arcball and dragging the cursor around the arcball causes the view to move around an axis that extends through the center of the arcball, perpendicular to the screen. This is called a *roll*.

**Horizontal Ellipse** When you move the cursor over one of the small circles on the left or right side of the arcball, it becomes a horizontal ellipse. Clicking and dragging from either of these points rotates the view around the vertical or *Y* axis through the middle of the arcball.



**Vertical Ellipse** When you move the cursor over one of the small circles on the top or bottom of the arcball, it becomes a vertical ellipse. Clicking and dragging from either of these points rotates the view around the horizontal or *X* axis through the middle of the arcball.



# 3DMESH

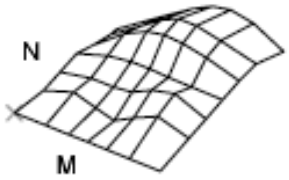
## Quick Reference

Creates a free-form polygon mesh

Draw ➤ Modeling ➤ Meshes ➤ 3D mesh
**3dmesh**
Enter size of mesh in M direction: *Enter a value between 2 and 256*
Enter size of mesh in N direction: *Enter a value between 2 and 256*

**NOTE** 3DMESH is designed primarily for programmers. Other users should use the *3D* command.

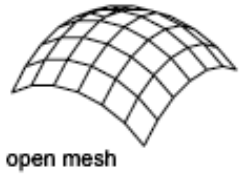
A polygon mesh is defined by a matrix, the size of which is determined by *M* and *N* size values. *M* times *N* equals the number of vertices that you must specify.



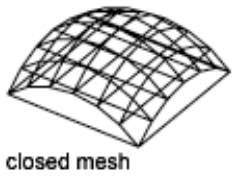

Specify location for vertex (0, 0): *Enter a 2D or 3D coordinate*

The location of each vertex in the mesh is defined by *m* and *n*, the row and column indices of the vertex. Defining vertices begins with vertex (0,0). You must supply the coordinate locations for each vertex in row *m* before specifying vertices in row *m* + 1.

Vertices may be any distance from each other. The *M* and *N* orientation of a mesh depends on the position of its vertices.




3DMESH polygon meshes are always open in both *M* and *N* directions. You can close a mesh with *PEDIT*.



# 3DMOVE

## Quick Reference

Displays the move grip tool in a 3D view and moves objects a specified distance in a specified direction

Modeling 
Modify ➤ 3D Operations ➤ 3D MoveAt the Command prompt, enter 3dmove.
**3dmove**
3D Make panel, 3D Move
Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify base point or [Displacement (page 42)] <Displacement>: *Specify a base point or enter* **d**
Specify second point or <use first point as displacement>: *Specify a point or press* ENTER

If you are working in a viewport with 2D Wireframe set as the visual style, 3DMOVE temporarily changes the visual style to 3D Wireframe for the duration of the command.

The move grip tool is displayed at the specified base point. For more information about using the move grip tool, see Use the Move Grip Tool to Modify Objects.

Click an axis handle to constrain the movement to an axis.

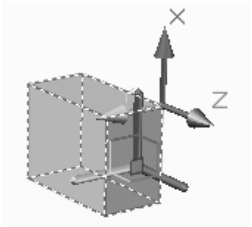
The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.

If you press ENTER at the Specify Second Point prompt, the first point is interpreted as a relative *X,Y,Z* displacement. For example, if you specify **2,3** for the base point and press ENTER at the next prompt, the objects move 2 units in the *X* direction and 3 units in the *Y* direction from their current position.

### Realigning the UCS

You can press CTRL+D to turn on the dynamic UCS to realign the grip tool as you move the pointer over faces, line segments, and polyline segments. The grip tool orients the workplane depending on which edge of the face the

pointer crosses. You can click to place the grip tool (which will constrain the direction of the move operation). Specified coordinates are relative to this workplane.




Pressing CTRL+D again to turn off dynamic UCS before placing the grip tool reverts the grip tool's orientation to match the static UCS.

#### Displacement

Specify displacement <*last value*>: *Enter coordinates to represent a vector*

Places the move grip tool at the origin (0,0,0). The coordinate values that you enter specify a relative distance and direction.

# 3DORBIT

## Quick Reference

Controls the interactive viewing of objects in 3D


3D Navigate
View ➤ Orbit ➤ Constrained OrbitAt the Command prompt, enter 3dorbit.
Press SHIFT and click the mouse wheel to temporarily enter 3D Orbit mode.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigations Modes  Constrained Orbit (1).
**3dorbit**
3D Navigate panel, Constrained Orbit flyout

3DORBIT activates a 3D Orbit view in the current viewport. You can view your entire drawing or select one or more objects before starting the command.

When 3DORBIT is active, the target of the view stays stationary and the camera location, or point of view, moves around the target. However, it appears as if

the 3D model is turning as the mouse cursor is dragged In this way, you can specify any view of the model.

The 3D Orbit cursor icon appears. If you drag the cursor horizontally, the camera moves parallel to the XY plane of the world coordinate system (WCS). If you drag the cursor vertically, the camera moves along the *Z* axis.

**NOTE** You cannot edit objects while the 3DORBIT command is active.

While the command is active, you can access additional 3DORBIT options and modes from a shortcut menu by right-clicking in the drawing area, or choosing buttons on the 3D Navigation toolbar. See 3DOrbit Shortcut Menu (page 43).

# 3DOrbit Shortcut Menu

## Quick Reference

When the 3DORBIT command (or any 3D navigation command or mode) is active, you can access the options on the 3D Orbit shortcut menu. To access the 3D Orbit shortcut menu, right-click in the 3D Orbit view.

### Current Mode: Current

Displays the current mode.

### Other Navigation Modes

Choose one of the following 3D navigation modes:

- **Constrained Orbit (1)**. Constrains orbiting to the XY plane or the Z direction.
- **Free Orbit (2)**. Allows orbiting in any direction, without being constrained to the XY plane or the Z direction. See *3DFORBIT*.
- **Continuous Orbit (3)**. Changes the cursor to a sphere with two continuous lines encircling it and enables you to set the objects into continuous motion. See *3DCORBIT*.
- **Adjust Distance (4)**. Simulates moving the camera closer to the object or farther away. See *3DDISTANCE*.

- **Swivel (5)**. Changes the cursor to an arched arrow and simulates the effect of swiveling the camera. See *3DSWIVEL*.
- **Walk (6)**. Changes the cursor to a plus sign and enables you to "walk through" a model at a fixed height above the XY plane, by dynamically controlling the location and target of the camera. See *3DWALK*.
- **Fly (7)**. Changes the cursor to a plus sign and enables you to "fly through" a model without being restricted to a fixed height above the XY plane. See *3DFLY*.
- **Zoom (8)**. Changes the cursor to a magnifying glass with plus (+) and minus (-) sign and simulates moving the camera closer to an object or farther away. Works like the Adjust Distance option. See *3DZOOM*.
- **Pan (9)**. Changes the cursor to a hand cursor and moves the view in the direction that you drag. See *3DPAN*.

**TIP** You can switch to any mode by using the shortcut menu or by entering the number displayed after its name.

### Enable Orbit Auto Target

Keeps the target point on the objects you are viewing rather than on the center of the viewport. This feature is turned on by default.

### Animation Settings

Opens the Animation Settings dialog box (page 54), where you can specify settings for saving an animation file.

### Zoom Window

Changes the cursor to a window icon so that you can select a specific area to zoom in on. When the cursor changes, click a starting point and end point to define the zoom window. The drawing is zoomed in and focused on the area you selected.

### Zoom Extents

Centers the view and sizes it to display all objects.

### Zoom Previous

Displays the previous view.

### Parallel

Displays objects so that two parallel lines in a drawing never converge. The shapes in your drawing always remain the same and do not appear distorted when they are closer.

### Perspective

Displays objects in perspective so that all parallel lines converge at one point. Objects appear to recede into the distance, and parts of the objects appear larger and closer to you. The shapes are somewhat distorted when the object is very close. This view correlates more closely to what your eyes see. See *PERSPECTIVE*.

### Reset View

Resets the view back to the view that was current when you first started 3DORBIT.

### Preset Views

Displays a list of predefined views such as Top, Bottom, and SW Isometric. Choose a view from the list to change the current view of your model.

### Named Views

Displays a list of named views in the drawing. Choose a named view from the list to change the current view of your model.

### Visual Styles

Provides methods for shading objects. For more information about visual styles, see Use a Visual Style to Display Your Model.

- **3D Hidden**. Displays the objects using 3D wireframe representation and hides lines representing back faces.
- **3D Wireframe**. Displays the objects using lines and curves to represent the boundaries.

- **Conceptual**. Shades the objects and smooths the edges between polygon faces. The effect is less realistic, but it can make the details of the model easier to see.
- **Realistic**. Shades the objects and smooths the edges between polygon faces.

#### Visual Aids

Provides aids to visualizing the objects.

- **Compass**. Draws a 3D sphere composed of three lines representing the *X*, *Y*, and *Z* axes.
- **Grid**. Displays a two-dimensional array of lines similar to graph paper. This grid is oriented along the *X* and *Y* axes.

  **NOTE** Before starting *3DORBIT*, you can use the *GRID* command to set system variables that control the grid display. The number of major grid lines corresponds to the value you set using the Grid Spacing option of the GRID command, which is stored in the *GRIDUNIT* system variable. Ten horizontal lines and ten vertical lines are drawn between the major lines.

- **UCS Icon**. Displays a shaded 3D UCS icon. Each axis is labeled *X*, *Y*, or *Z*. The *X* axis is red, the *Y* axis is green, and the *Z* axis is blue.

# 3DORBITCTR

## Quick Reference

Sets the center of rotation in 3D Orbit view

#### 3dorbitctr

Starts 3D Orbit view and uses a center of rotation that you specify with your pointing device. If you specify a point outside the current view, 3DORBITCTR ignores the specified point and uses the default center of rotation.

3DORBITCTR overrides the AutoTarget option in the *3DORBIT* command.

# 3DPAN

## Quick Reference

When a drawing is in a Perspective view, starts the interactive 3D view and enables you to drag the view horizontally and vertically

3D Navigation 
View ➤ PanDoes not exist in the menus.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigation Modes ➤ Pan (9).
**3dpan**
3D Navigate panel, Pan

Moves in the direction that you drag. You can drag the view vertically, horizontally, or diagonally. 3DPAN changes the cursor to a hand cursor.

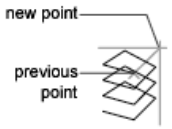
You can view your entire drawing or select objects before entering 3DPAN.

# 3DPOLY

## Quick Reference

Creates a polyline of line segments in 3D space

Draw ➤ 3D PolylineAt the Command prompt, enter 3dpoly.
**3dpoly**
Specify start point of polyline: *Specify a point (1)*
Specify endpoint of line (page 47) or [Undo (page 48)]: *Specify a point or enter an option*
Specify endpoint of line or [Undo]: *Specify a point or enter an option*
Specify endpoint of line or [Close (page 48)/Undo]: *Specify a point or enter an option*
**Endpoint of Line** Draws a straight line from the previous point to the specified new point. The prompt is repeated until you press ENTER to end the command.




**Undo** Deletes the last line created. You can continue drawing from the previous point.


**Close** Draws a closing line from the endpoint back to the first point, and then ends the command. To be closed, a 3D polyline must contain at least two lines.


# 3DROTATE

## Quick Reference

Displays the rotate grip tool in a 3D view and revolves objects around a base point

Modeling 
Modify ➤ 3D Operations ➤ 3D RotateAt the Command prompt, enter 3drotate.
**3drotate**
3D Make panel, 3D Rotate
Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify base point: *Specify a base point*

If you are working in a viewport with 2D Wireframe set as the visual style, 3DROTATE temporarily changes the visual style to 3D Wireframe for the duration of the command.

The rotate grip tool is displayed at the specified base point. For more information about using the rotate grip tool, see Use the Rotate Grip Tool to Modify Objects.

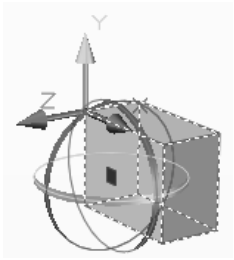
Pick a rotation axis: *Click an axis handle to select the rotation axis*

Enter a value for the rotation or specify the angle start point and endpoint.

Specify angle start point: *Specify a point*
Specify angle endpoint: *Specify a point*

### Realigning the UCS

You can press CTRL+D to turn on the dynamic UCS to realign the grip tool as you move the pointer over faces, line segments, and polyline segments. The grip tool orients the workplane depending on which edge of the face the pointer crosses. You can click to place the grip tool (which will constrain the direction of the move operation). Specified coordinates are relative to this workplane.




Pressing CTRL+D again to turn off dynamic UCS before placing the grip tool reverts the grip tool's orientation to match the static UCS.

# 3DSIN

## Quick Reference

Imports a 3D Studio (3DS) file

Insert ➤ 3D StudioAt the Command prompt, enter 3dsin.
**3dsin**

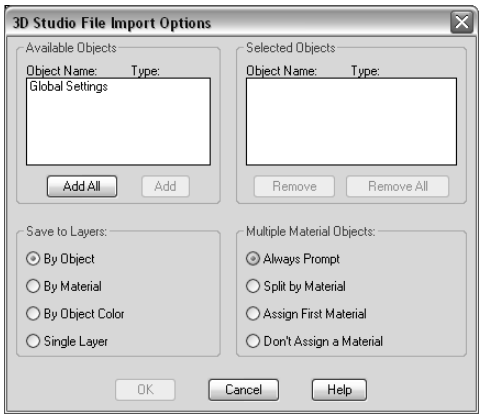
The 3D Studio File Import dialog box (a standard file selection dialog box (page 931)) is displayed. After you choose a file to import, the 3D Studio File Import Options dialog box (page 50) is displayed.

# 3D Studio File Import Options Dialog Box

## Quick Reference

Insert ➤ 3D StudioAt the Command prompt, enter 3dsin.
**3dsin**

Lists available and selected 3D objects, saves 3D objects to layers, and handles objects that use multiple materials.




### Available Objects

Displays the names of all objects in the 3D Studio® (3DS) file. You can select up to 70 objects.

**Object Name and Type** Displays the type and assigned name of each object.

**Add All** Adds all of the objects in the Available Objects list to the Selected Objects list.

**Add** Adds the objects currently selected in the Available Objects list to the Selected Objects list.

### Selected Objects

Displays the selected 3D Studio objects to import.

**Object Name and Type** Displays the type and assigned name of each object.

**Remove** Removes objects selected in the Selected Objects list and returns them to the Available Objects list.

**Remove All** Removes all objects from the Selected Objects list and returns them to the Available Objects list.

### Save to Layers

Controls how 3D Studio objects are assigned to layers in the drawing.

**By Object** Creates a layer for each object in the 3D Studio file and places the object on that layer. The name of the layer is the same as the name of the object.

**By Material** Creates a layer for each material in the 3D Studio file and places objects to which that material is attached on that layer. The name of the layer is the same as the name of the material.

**By Object Color** Creates a layer for each object color in the 3D Studio file. Each 3D Studio object is placed on the layer corresponding to its color. The name of the layer is COLOR*nn*, where *nn* is the 3D Studio color index. If the 3D Studio file contains objects with no color, these objects are placed on a layer called COLORNONE.

**Single Layer** Creates a single layer called AVLAYER and places all objects on that layer.

### Multiple Material Objects

3D Studio assigns materials by face, element, or object. AutoCAD assigns materials by object only. When AutoCAD encounters a 3D Studio object assigned multiple materials, AutoCAD must find a way to handle the assignment.

### Always Prompt

Displays the Material Assignment Alert dialog box for each object with multiple materials. The dialog box displays the name of the object. The options you select determine how 3DSIN handles the assignment. The options are as follows:

**Split Object by Materials** Splits the object into multiple objects, one for each material. This preserves the material assignments that were made in the 3D Studio file. It also increases the complexity of the drawing geometry.

**Assign First Material** Assigns the first material assigned to each multiple-material object to the entire object. See the 3D Studio documentation for an explanation of how the first-assigned material is determined.

**Select a Material** Assigns one of the materials that was assigned to the object in the 3D Studio file to the entire object. Select the material from the list below this option, or choose NONE to revert to the AutoCAD default material.

#### Split by Material

Splits all objects with multiple materials into multiple objects, one for each material. This preserves the material assignments that were made in the 3D Studio file. It also increases the complexity of the drawing geometry.

#### Assign First Material

Assigns the first material assigned to each multiple-material object to the entire object. See the 3D Studio documentation for an explanation of how the first-assigned material is determined.

#### Don't Assign a Material

Assigns no material to each multiple-material object. This option loses all material assignment information but preserves the 3D Studio geometry. The object reverts to the AutoCAD default material.

# 3DSOUT

## Quick Reference

Obsolete

For best results, it is recommended that you import DWG files into 3D Studio Max instead.

**3dsout**

# 3DSWIVEL

## Quick Reference

Changes the target of the view in the direction that you drag

3D Navigation 

View ➤ Camera ➤ SwivelAt the Command prompt, enter 3dswivel.

Press CTRL and click the mouse wheel to temporarily enter 3DSWIVEL mode.

Start any 3D navigation command, right-click in the drawing area, and click Other Navigations Modes  Swivel (5).

**3dswivel**

3D Navigate panel, Swivel flyout

Simulates panning with a camera in the direction that you drag. The target of the view changes. You can swivel the view along the *XY* plane or along the *Z* axis.

# 3DWALK

## Quick Reference

Interactively changes the view of a 3D drawing so that you appear to be walking through the model

3D Navigation 

View ➤ Walk and Fly ➤ Walk

Start any 3D navigation command, right-click in the drawing area, and click Other Navigation Modes ➤ Walk (6).

**3dwalk**

3D Navigate panel, Walk flyout

3DWALK activates a walk mode in the current viewport. On the keyboard, use the four arrow keys or the W (forward), A (left), S (back), and D (right) keys to determine the direction of your walk. To specify the direction of the view, you drag the mouse in the direction you want to look. For more information, see Walk and Fly Navigation Mappings Dialog Box (page 56).

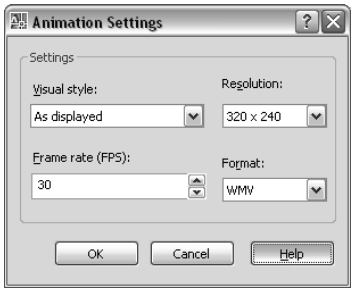
By default, the Position Locator window (page 54) opens and displays your position in the drawing from a top view.

To control walk and fly settings, use the Walk and Fly Settings dialog box. (page 1546)To specify settings for animating a 3D walk or fly, see Animation Settings Dialog Box (page 54).

# Animation Settings Dialog Box

## Quick Reference

Specifies settings for recording an animation of 3D navigation.




**Settings** Determines the settings for an animation file.

**Visual Style** Displays a list of visual styles and render presets that you can apply to an animation file.

**Resolution** Displays a list of resolutions. The default value is 320 x 240.

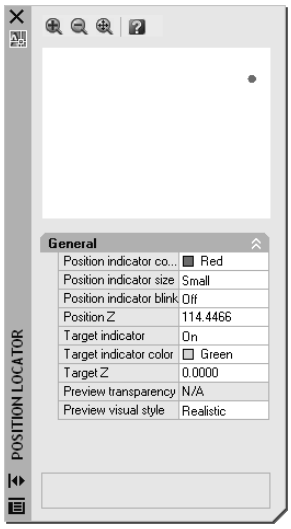
**Frame Rate (FPS)** Specifies a frame rate (in seconds). Values are between 1 and 60. The default value is 30.

**Format** Specifies a list of animation output formats. Available formats are AVI, MPG, WMV, and MOV.

# Position Locator Window

## Quick Reference

Displays the top-view position of a 3D model when in a walk or fly navigation.




**Zoom In** Zooms in to the display in the Position Locator window.

**Zoom Out** Zooms out from the display in the Position Locator window.

**Zoom Extents** Zooms to the extents of the display in the Position Locator window.

**Preview** Displays your current location in the model. You can drag the position indicator to change your position. You can also drag the target indicator to change the direction of the view.

**Position Indicator Color** Sets the color of the dot that shows your current position.

**Position Indicator Size** Sets the size of the indicator. You can choose Small, Medium, or Large.

**Position Indicator Blink** Turns the blinking effect on or off.

**Target Indicator** Displays an indicator that shows the target of the view.

**Target Indicator Color** Sets the color of the target indicator.

**Preview Transparency** Sets the transparency of the preview window. You can choose a value from 0 to 95.

**Preview Visual Style** Sets the visual style of the preview.

# Walk and Fly Navigation Mappings Dialog Box

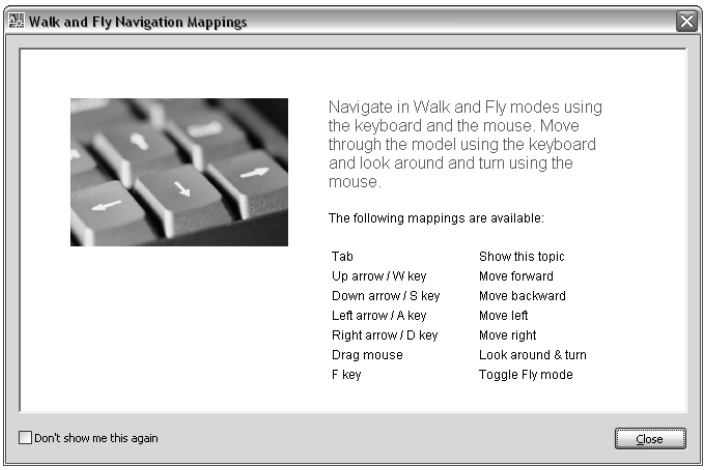
## Quick Reference

The Walk and Fly Navigation Mappings dialog box displays the keyboard and mouse controls that control walk and fly modes.

Press TAB to toggle the display of the dialog box. You can specify whether to display the instructions, and how often, in the Walk and Fly Settings dialog box (page 1546).

The controls are as follows:

- **Up arrow/W key**. Moves the camera forward so that you appear to be walking (or flying) forward.
- **Down arrow/S key**. Moves the camera backward.
- **Left arrow/A key**. Moves the camera to the left, so that you appear to be moving to the left.
- **Right arrow/D key**. Moves the camera to the right.
- **Drag mouse**. Sets the target so that you look or turn.
- **F key**. Toggles between fly (*3DFLY*) and walk (*3DWALK*) modes.



# 3DZOOM

## Quick Reference

Zooms in and out in a perspective view

3D Navigation 
View ➤ ZoomDoes not exist in the menus.
Scroll the mouse wheel to zoom in and out.
Start any 3D navigation command, right-click in the drawing area, and click Other Navigation Modes ➤ Zoom (8).
**3dzoom**
3D Navigate panel, Zoom

In a parallel view, *3DZOOM* displays the following prompt:

Press ESC or ENTER to exit, or right-click to display shortcut-menu.

Zooming in a perspective view simulates moving the camera closer to the target or farther away. Objects appear closer or farther away, but the position of the camera does not change.

Enter option [All (page 57)/Extents (page 57)/Window (page 57)/Previous (page 57)] <real time (page 57)>:
**All** Zooms to display the entire drawing.

**Extents** Zooms to display the drawing extents and results in the largest possible display of all the objects

**Window** Zooms to display an area specified by two points of a rectangular window.

**Previous** Zooms to display the previous view.

**Real Time** Using the pointing device, zooms interactively.

# A Commands

# 2

**In this chapter**

- ATTEDIT
- ATTEXT
- ATTIPEDIT
- ATTREDEF
- ATTSYNC
- AUDIT
- AUTOPUBLISH

# ABOUT

## Quick Reference

Displays information about AutoCAD

Help ➤ AboutAt the Command prompt, enter about.
**about** (or '**about** for transparent use)

Copyright information and product information is displayed. Product information includes the version number and service pack, serial number, license type and expiration date, and the text of the license agreement. You can save the product information as a text file.

# ACISIN

## Quick Reference

Imports an ACIS file and creates a body object, solid, or region in the drawing

Insert ➤ ACIS FileAt the Command prompt, enter acisin.
**acisin**

The Select ACIS File box is displayed. Select the file to import in the File Name list. The SAT (ASCII) ACIS file is imported into the drawing.

**NOTE** ACISIN imports SAT files up to ACIS version 7.0.

# ACISOUT

## Quick Reference

Exports a body object, solid, or region to an ACIS file

**acisout**

**NOTE** When exchanging SAT files to earlier versions of AutoCAD, you need to set the *ACISOUTVER* system variable to the ACIS version used for that release. For example, to export SAT files to AutoCAD Release 14, set ACISOUTVER to 16.

Select objects: *Use an object selection method*

Selected objects that are not solids or regions are ignored, and the Create ACIS File dialog box is displayed. Enter the name of the file you want to create. The selected objects are exported to an ASCII file.

# ADCCLOSE

## Quick Reference

Closes DesignCenter

**adcclose**

Closes the DesignCenter™ window.

# ADCENTER

## Quick Reference

Manages and inserts content such as blocks, xrefs, and hatch patterns

Standard 
Tools ➤ Palettes ➤ DesignCenterAt the Command prompt, enter adcclose.
**adcenter**

DesignCenter (page 62)is displayed.

# DesignCenter Window

## Quick Reference

Standard 
Tools ➤ Palettes ➤ DesignCenter Does not exist in the menus.

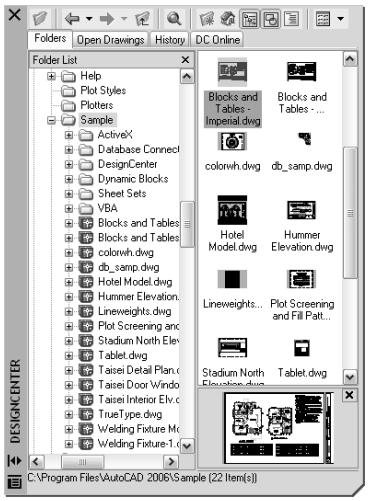
**adcenter**

Browses, finds, and previews content, and inserts content, which includes blocks, hatches, and external references (xrefs).

Use the buttons in the toolbar at the top of DesignCenter for display and access options.

When you click the Folders tab or the Open Drawings tab, the following two panes are displayed from which you can manage drawing content:

- Content area (right pane) (page 63)
- Tree view (left pane) (page 67)




## Content Area (DesignCenter)

Displays the content of the "container" currently selected in the tree view. A container is a network, computer, disk, folder, file, or web address (URL) containing information accessible by DesignCenter. Depending on the container selected in tree view, the content area typically displays the following:

- Folders containing drawings or other files
- Drawings
- Named objects contained in drawings (named objects include blocks, xrefs, layouts, layers, dimension styles, and text styles)

- Images or icons representing blocks or hatch patterns
- Web-based content
- Custom content developed by third-party applications

From the content area, you can insert blocks or hatch patterns or attach external references in a drawing by dragging, by double-clicking, or by right-clicking and choosing Insert Block, Attach Xref, or Copy. You can drag or right-click to add other content to drawings, such as layers, dimension styles, and layouts. You can drag blocks and hatches from DesignCenter to tool palettes.

**NOTE** You can access relevant content area and tree view options on a shortcut menu by right-clicking in the tree view or the content area.

**Load** 

Displays the Load dialog box (a standard file selection dialog box (page 931)). Use Load to navigate to files on local and network drives or on the Web, and then to select content to load in the content area.

**Back** 

Returns to the most recent location in the history list.

**Forward** 

Returns to the next later location in the history list.

**Up** 

Displays the contents of the container one level above the current container.

**Stop (DC Online tab)** 

Stops the current transfer.

**Reload (DC Online tab)** 

Reloads the current page.

**Search** 

Displays the Search dialog box (page 68), where you can specify search criteria to locate drawings, blocks, and nongraphical objects within drawings.

Search also displays custom content saved on your desktop.

**Favorites** 

Displays the contents of the *Favorites* folder in the content area. The *Favorites* folder contains shortcuts to items you access often. You can add items to *Favorites* either by right-clicking the content area or right-clicking an item in the tree view, and then clicking Add to Favorites. To delete an item from *Favorites*, use the Organize Favorites option on the shortcut menu and then use the Refresh option on the shortcut menu.

**NOTE** The *DesignCenter* folder is automatically added to *Favorites.* This folder contains drawings with discipline-specific blocks that you can insert in drawings.

**Home** 

Returns DesignCenter to your home folder. On installation, the home folder is set to *...\Sample\DesignCenter*. Change the home folder using the shortcut menu in the tree view.

**Tree View Toggle** 

Displays and hides the tree view. Hide the tree view if you need more space in your drawing area. When the tree view is hidden, you can use the content area to navigate to containers and to load content.

The Tree View Toggle button is not available while you're using the History list in the tree view.

**Preview** 

Displays and hides a preview of the selected item in a pane below the content area. If there is no preview image saved with the selected item, the Preview area is empty.

**Description** 

Displays and hides a text description of the selected item in a pane below the content area. If a preview image is also displayed, the description is displayed below it. If there is no description saved with the selected item, the Description area is empty.

**Views** 

Provides different display formats for the content that is loaded in the content area. You can select a view from the Views list or click the Views button repeatedly to cycle through the display formats. The default view varies for the type of content currently loaded in the content area.

**Large Icon** Displays the names of the loaded content in large icon format.

**Small Icon** Displays the names of the loaded content in small icon format.

**List View** Displays the names of the loaded content in a list.

**Detail View** Displays additional information about the loaded content. You can sort the items by name, size, type, and other properties, depending on the type of content that is loaded in the content area.

### Refresh (Shortcut Menu Only)

Refreshes the display in the content area to reflect any changes you have made. Right-click the content area background and click Refresh on the shortcut menu.

### Tree View (DesignCenter)

Displays the hierarchy of files and folders on your computer and network drives, a list of open drawings, custom content, and a history of the last locations you accessed. Select an item in the tree view to display its contents in the content area.

---

**NOTE** In the *sample\designcenter* folder are drawings containing discipline-specific blocks that you can insert in drawings. These drawings are called symbol library drawings.

---

Use the buttons in the toolbar at the top of DesignCenter to access tree view options.

**Folders** Displays the hierarchy of files and folders on your computer and network drives, including My Computer and Network Neighborhood.

You can use ADCNAVIGATE to navigate to a specific file name, directory location, or network path in the DesignCenter tree view. See *ADCNAVIGATE*.

**Open Drawings** Displays all drawings open in the current work session, including drawings that are minimized.

**History** Displays a list of the files that you opened most recently in DesignCenter. With the history displayed, right-click a file to display information about the file or to delete the file from the History list.

**DC Online** Accesses the DesignCenter Online web page. When you establish a web connection, two panes are viewed on the Welcome page. The left side displays folders containing symbol libraries, manufacturer sites, and additional content libraries. When a symbol is selected, it is displayed on the right side and can be downloaded into your drawing.

# Search Dialog Box

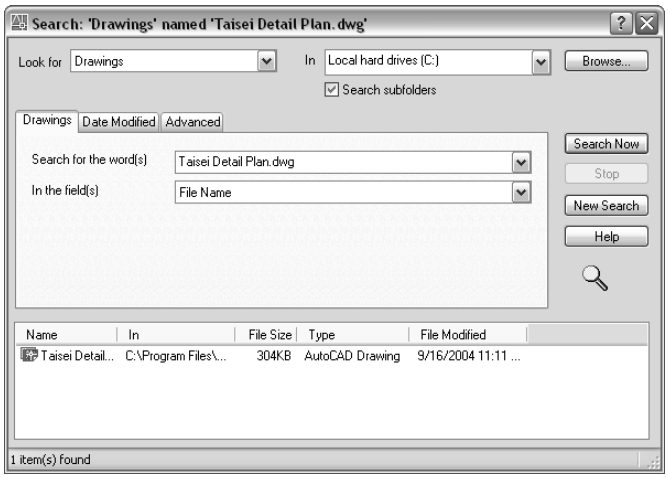
## Quick Reference

Standard

Tools ➤ Palettes ➤ DesignCenterDoes not exist in the menus.

**adcenter**

Searches for content such as drawings, hatch patterns, and blocks.




**Look For** Specifies the type of content to search for. The content type you specify determines which tabs are displayed in the Search dialog box and the search fields it provides. The Date Modified and Advanced tabs are displayed only when the Drawings option is selected in Look For.

**In** Specifies a search path name. To enter multiple paths, separate them with semicolons. Use Browse to select a path from a tree view list.

**Browse** Displays a tree view in the Browse for Folder dialog box, in which you can specify the drives and folders to search.

**Search Subfolders** Includes subfolders in the search path.

**Search Now** Starts the search based on criteria you specify.

**Stop** Stops the search and displays the accumulated results in the Search Results panel.

**New Search** Clears the Search For the Word(s) box and places the cursor in the box.

**Search Results Panel** Displays the results of the search in resizable columns at the bottom of the window. Double-click an item to load it into DesignCenter.

### Search Tabs (Search Dialog Box)

Display search fields relevant to the type of content specified in the Look For list. The name of the tab reflects the content type. You can use wild-card characters to extend or limit your search patterns.

**Search for the Word(s)** Specifies the text string to look for in the field you specify. Use the asterisk and question mark wild-card characters to broaden your search.

- ** (Asterisk):* Matches any string and can be used anywhere in the search string.
- *? (Question mark):* Matches any single character; for example, ?BC matches ABC, 3BC, and so on.

This option is available on the Drawings tab and, when available, the Custom Content tab.

**In the Field(s)** Specifies the property fields to search. For drawings, all fields other than File Name are derived from the information entered in the Drawing Properties dialog box (*DWGPROPS*).

This option is available on the Drawings tab and the Custom Content tab.

---

**NOTE** Custom content developed by third-party applications may not provide fields for searches using the Search dialog box.

---

**Search for the Name** Specifies the name of the drawing, block, layer, xref, or other content type that you want to find.

This option is available on all tabs except Drawings and Custom Content.

### Date Modified Tab (Search Dialog Box)

Finds content that was created or modified during a certain time period.

**All Files** Finds all the files that match the criteria you specify on other tabs, regardless of creation date or date modified. This is the default setting.

**Find All Files Created or Modified** Finds files created or modified during a specific time period. The search locates files that match the criteria you specify on this and other tabs.

**Between** Searches for files created or modified between the dates you specify.

**During the Previous Month(s)** Searches for files created or modified within the number of months you specify.

**During the Previous Day(s)** Searches for files created or modified within the number of days you specify.

#### Advanced Tab (Search Dialog Box)

Finds content within drawings; this tab is available only when you select Drawings in Look For.

**Containing** Specifies the type of text in a drawing that you want to search for. For example, you can search for text contained within a block attribute, such as a manufacturer's catalog number.

**Containing Text** Specifies the text you want to search for.

**Size Is** Specifies a minimum or maximum file size. Select At Least or At Most, and enter a value in KB.

# ADCNAVIGATE

## Quick Reference

Loads a specified DesignCenter drawing file, folder, or network path

**adcnavigate**
Enter pathname <*current*>:

The path or drawing file name you specify is loaded in the tree view of the DesignCenter Folders tab.

At the prompt, you can enter a path in any of the following formats:

- Folder path: *c:\project files\electrical*
- Folder path and a file name: *c:\project files\electrical\circuit2.dwg*

- UNC (universal naming convention) network path: *server1**campus_project*

DesignCenter is displayed, with the Folder tab active, and the path or drawing file that you specified is loaded.

ADCNAVIGATE supports remote domains or workgroups if you have the target domain or workgroup mapped to a drive letter.

# ALIGN

## Quick Reference

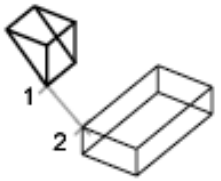
Aligns objects with other objects in 2D and 3D

Modify ➤ 3D Operations ➤ AlignAt the Command prompt, enter align.
**align**
Select objects: *Select the objects to align and press* ENTER
**ALIGN Using One Pair of Points** Specify first source point: *Specify a point (1)*

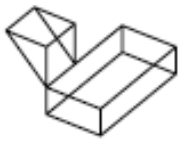
Specify first destination point: *Specify a point (2)*

Specify second source point: *Press* ENTER

When you select only one source point and destination point pair, the selected objects move in 2D or 3D from the source point (1) to the destination point (2).



two points specified


result

**ALIGN Using Two Pairs of Points** Specify first source point: *Specify a point (1)*

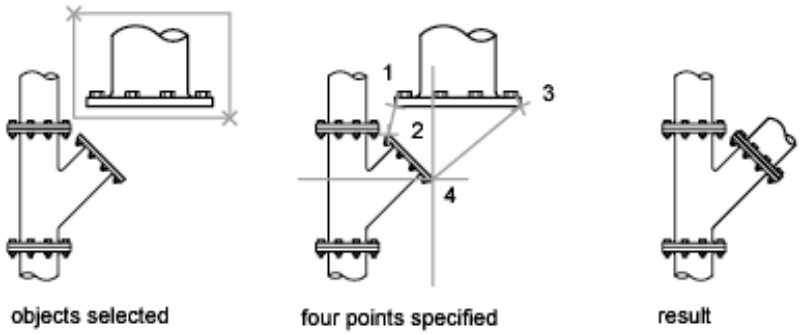
Specify first destination point: *Specify a point (2)*

Specify second source point: *Specify a point (3)*

Specify second destination point: *Specify a point (4)*

Specify third source point: *Press***ENTER**

Scale objects based on alignment points [Yes/No] <No>: *Enter***y***or press***ENTER**




When you select two point pairs, you can move, rotate, and scale the selected objects in 2D or 3D to align with other objects.

The first set of source and destination points defines the base point for the alignment (1, 2). The second set of points defines the angle of rotation (3, 4).

After you enter the second set of points, you are prompted to scale the object. The distance between the first and second destination points (2, 4) is used as the reference length to which the object is scaled. Scaling is available only when you are aligning objects using two point pairs.

---

**NOTE** If you use two source and destination points to perform a 3D alignment on nonperpendicular working planes, you get unpredictable results.

---

**ALIGN Using Three Pairs of Points** Specify first source point: *Specify a point (1)*

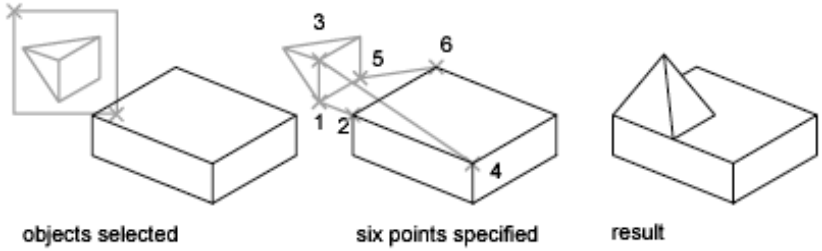
Specify first destination point: *Specify a point (2)*

Specify second source point: *Specify a point (3)*

Specify second destination point: *Specify a point (4)*

Specify third source point: *Specify a point (5)*

Specify third destination point: *Specify a point (6)*



When you select three point pairs, you can move and rotate the selected objects in 3D to align with other objects.

The selected objects move from the source point (1) to the destination point (2).

The selected object is rotated (1 and 3) so that it aligns with the destination object (2 and 4).

The selected object is then rotated again (3 and 5) so that it aligns with the destination object (4 and 6).

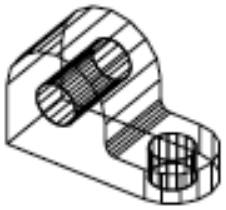
# AMECONVERT

## Quick Reference

Converts AME solid models to AutoCAD solid objects

**ameconvert**
Select objects: *Use an object selection method*

The objects you select must be Advanced Modeling Extension (AME) Release 2 or 2.1 regions or solids. All other objects are ignored.



Because of increased accuracy in the new solid modeler, AME models may look slightly different after conversion. This difference is noticeable where the previous version of the solid modeler identified the surfaces of two different shapes as so close as to be considered in the same plane. The new solid modeler's finer tolerance may interpret these surfaces as being slightly offset. This phenomenon is most apparent with aligned features such as fillets, chamfers, and through-holes.

Holes might become blind holes when the new modeler, with its much finer approximation capability, interprets what was once a through-hole as being slightly less wide than the solid. Typically, the length of the remaining solid material is the difference between the tolerance of the previous modeler and that of the new modeler.

Likewise, updated fillets or chamfers can occasionally be placed slightly below the surface, creating a hole through the solid, leaving the original shape unaltered. Also, drawing fillets or chamfers slightly above the original surface creates an uneven transition between the solid and the fillet or chamfer.

# ANIPATH

### Quick Reference

Saves an animation along a path in a 3D model

View ➤ Motion Path AnimationsAt the Command prompt, enter anipath.
**anipath**

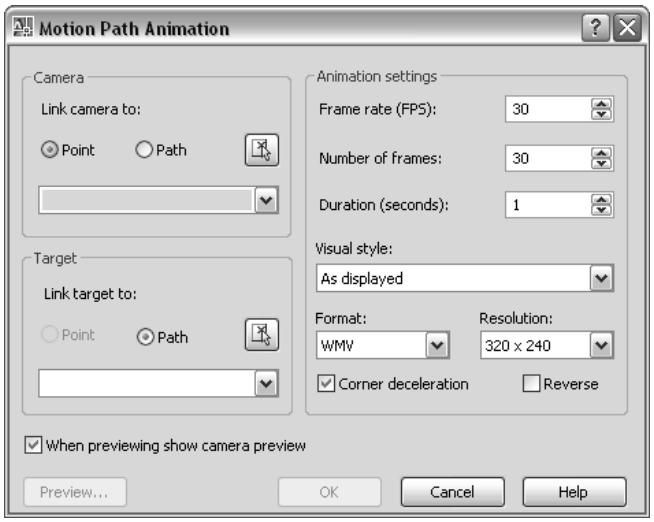
The Motion Path Animation dialog box (page 74) is displayed.

## Motion Path Animation Dialog Box

### Quick Reference

View ➤ Motion Path AnimationsAt the Command prompt, enter anipath.
**anipath**

Specifies settings for a motion path animation and creates an animation file.




### Camera

**Link Camera To** Links a camera to a static point or a motion path in a drawing.

**Point** Links a camera to a static point in a drawing.

**Path** Links a camera to a motion path in the drawing.

**Pick Point/Select Path** Selects either the point where a camera is located or the path along which a camera travels, depending on whether Point or Path was selected.

**Point/Path List** Displays a list of named points or paths to which you can link a camera. To create a path, you can link a camera to a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

---

**NOTE** When you create a motion path, a camera is automatically created. If you delete an object that you specified as a motion path, the named motion path is also deleted.

---

### Target

**Link Target To** Links a target to a point or a path.

If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, you can link the target to either a point or a path.

**Point** If the camera is linked to a path, links the target to a static point in the drawing.

**Path** Links the target to a motion path in the drawing.

**Pick Point/Select Path** Selects either the point or a path for the target, depending on whether Point or Path was selected.

**Point/Path List** Displays a list of named points or paths to which you can link the target. To create a path, you can link a target to a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

### Animation Settings

Controls the output of the animation file.

**Frame Rate (FPS)** The speed at which the animation will run, measured in frames per second. Specify a value from 1 to 60. The default value is 30.

**Number of Frames** Specifies the total number of frames in the animation.

With the frame rate, this value determines the length of the animation. When you change this number, the Duration value is automatically recalculated.

**Duration (seconds)** Specifies the duration (in sections) of the animation.

When you change this number, the Number of Frames value is automatically recalculated.

**Visual Style** Displays a list of visual styles and render presets that you can apply to an animation file. See *VISUALSTYLES* and *RENDERPRESETS* for more information.

**Format** Specifies the file format for the animation.

You can save an animation to an AVI, MOV, MPG, or WMV file format for future playback. The MOV format is available only if Apple QuickTime Player is installed. The WMV format is available and is the default selection only if Microsoft Windows Media Player 9 or later is installed. Otherwise, AVI is the default selection.

**Resolution** Defines the width and height of the resulting animation in screen display units. The default value is 320 x 240.

**Corner Deceleration** Moves a camera at a slower rate as it turns a corner.

**Reverse** Reverses the direction of an animation.

**When Previewing Show Camera Preview** Enables the display of the Animation Preview dialog box so you can preview the animation before you save it.

**Preview** Displays a preview of the animation in the Animation Preview dialog box (page 77).

# Animation Preview Dialog Box

## Quick Reference

View ➤ Motion Path AnimationsAt the Command prompt, enter anipath.
**anipath**

Previews a motion path animation that you created with a motion path or 3D navigation.


**Preview** Displays a preview of the animation you set up either in the Motion Path Animation dialog box or when walking or flying through an animation and recording it from the dashboard.

**Play** Plays the animation preview. The Play button is disabled when an animation is already playing.

**Record** Records the animation, starting from the current frame displayed in the Preview region. All frames following the current frame are overwritten. The Overwrite Confirmation warning is displayed to confirm that you intend

to overwrite the existing frames. The Record button is disabled when an animation is playing.

**Pause** Pauses the animation at the current frame displayed in the Preview region. After pausing an animation, the Pause button is disabled.

**Save** Opens the Save As dialog box. You can save an animation to an AVI, MOV, MPG, or WMV file format for future playback. Once you save the animation, you are returned to the drawing. The Save button is disabled when an animation is playing.

**Visual Style** Specifies the visual style displayed in the Preview region. The visual style is initially set to Current, which is the visual style defined in the active viewport. Choose from a list of preset and user-defined visual styles.

**Slider** Moves frame-by-frame through the animation preview. You can move the slider to a specific frame in the animation. A tooltip displays the current frame and total number of frames in the animation.

# ANNOUPDATE

## Quick Reference

Updates existing objects to match the current properties of their styles

**annoupdate**

When a non-annotative text object is updated to an annotative text style, the object becomes annotative and supports the current . If the text style has a fixed Paper Height, the object is set to that height. If the text style's Paper Height is set to 0, the size of the text does not change. The Paper Height value is inferred by dividing the model height by the current annotation scale.

If an annotative object is updated to a non-annotative style, the object becomes non-annotative and any alternate are removed.

Select objects: *Use an object selection method*

# ANNORESET

## Quick Reference

Resets the location of all for an object to that of the current scale representation

**annoreset**
Select objects: *Use an object selection method*

# APERTURE

## Quick Reference

Controls the size of the object snap target box

**aperture** (or '**aperture** for transparent use)
Object snap target height (1-50 pixels) <current>: *Enter a value* (**1-50**) *or press* ENTER

Object snap applies only to objects inside or crossing the object snap target box. The *APBOX* system variable controls whether the object snap target box is displayed. The number of pixels you enter using APERTURE controls the size of the object snap box. The higher the number, the larger the target box. You can also change this setting in the Options dialog box, Drafting tab.


APERTURE controls the object snap target box, not the pickbox displayed at the Select Objects prompt. The object selection pickbox is controlled by the *PICKBOX* system variable.

# APPLOAD

## Quick Reference

Loads and unloads applications and defines which applications to load at startup

Tools ➤ Load ApplicationAt the Command prompt, enter appload.

**appload**

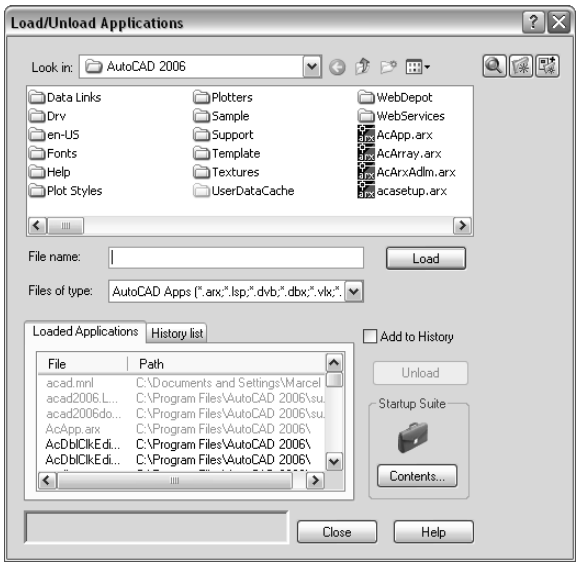
The Load/Unload Applications dialog box is displayed (page 80).

# Load/Unload Applications Dialog Box

## Quick Reference

Tools ➤ Load ApplicationAt the Command prompt, enter appload.
**appload**

Loads and unloads applications and specifies applications to be loaded at startup.




The options at the top of this dialog box are derived from the standard file selection dialog box (page 931). Following are descriptions of the additional options provided by the Load/Unload Applications dialog box:

**Load** Loads or reloads the applications that are currently selected either in the files list or on the History List tab. Load is unavailable until you select a file that you can load. ObjectARX, VBA, and DBX applications are loaded immediately, but LSP, VLX, and FAS applications are queued and then loaded when you close the Load/Unload Applications dialog box.

If you select a file that is already loaded, Load reloads the application when applicable. You cannot reload ObjectARX applications. In this case, you must first unload the ObjectARX application and then load it again. The Load option is also available from a shortcut menu by right-clicking a file on the History List tab.

**Loaded Applications** Displays an alphabetical list (by file name) of currently loaded applications. LISP routines are displayed in this list only if you loaded them in the Load/Unload Applications dialog box. You can drag files into this list from the files list or from any application with dragging capabilities, such as Microsoft® Windows® Explorer.

If you use the AutoCAD web browser to load an application, the web browser downloads the application to a temporary location on your machine. This is the location from which AutoCAD loads the application, as displayed in this list.

You can also unload certain applications from this list. See the Unload option for details. Files that you cannot unload are not available for selection.

**History List** Displays an alphabetical list (by file name) of applications that you previously loaded with Add To History selected. You can drag files into this list from the files list, or from any application with dragging capabilities, such as Windows Explorer. If Add To History is not selected when you drag files into this list, the dragged files are loaded but not added to the history list.

You can load and remove applications from this list, but to unload applications, you must use the Loaded Applications tab. See the Load, Unload, and Remove options.

**Add to History** Adds any applications that you load to the history list.

You may prefer to clear this option when loading applications with the AutoCAD web browser, because these applications are unavailable once the cache for the application's temporary location is emptied.

**Unload/Remove** Unloads the selected applications or removes them from the History List. Unload is available only when a file is selected on the Loaded Applications tab. Remove is available only when you select a file on the History List tab.

LISP applications cannot be unloaded, nor can ObjectARX applications that are not registered for unloading.

Note that Remove does not unload the selected application. The Remove option is also available from a shortcut menu by right-clicking an application on the History List tab.

**Startup Suite** Contains a list of applications that are loaded each time you start AutoCAD. You can drag application files from the files list, or from any application with dragging capabilities such as Windows Explorer, into the Startup Suite area to add them to the Startup Suite.

You cannot add applications that you load with the AutoCAD web browser to the Startup Suite.

**Contents** Displays the Startup Suite dialog box (page 82). You can also add files to the Startup Suite by clicking the Startup Suite icon or by right-clicking an application on the History List tab and clicking Add to Startup Suite on the shortcut menu.

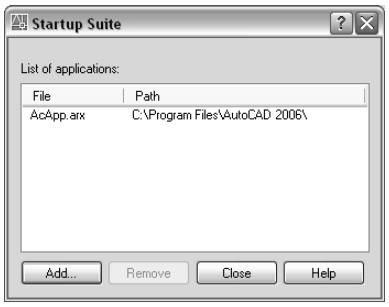
**Status Line** Displays messages that indicate the status of loading and unloading operations.

# Startup Suite Dialog Box

## Quick Reference

Tools ➤ Load ApplicationAt the Command prompt, enter appload.
**appload**

Adds and removes application files from the Startup Suite. These are the applications that are loaded each time you start AutoCAD.




**List of Applications** Displays an alphabetical list (by file name) of the application files to load at startup.

**Add** Displays the Add File to Startup Suite dialog box. You can use this dialog box to select files to add to the startup suite.

**Remove** Removes selected files from the Startup Suite.

# ARC

## Quick Reference

Creates an arc

Draw 

Draw ➤ ArcDoes not exist in the menus.

**arc**

Specify start point (page 83) of arc or [Center (page 85)]:*Specify a point, enter* **c***, or press ENTER to* start tangent to last line, arc, or polyline (page 86)

### Start Point

Specifies the starting point of the arc.

---

**NOTE** If you press ENTER without specifying a point, the endpoint of the last drawn line or arc is used and you are immediately prompted to specify the endpoint of the new arc. This creates an arc tangent to the last drawn line, arc, or polyline.

---

Specify second point of arc or [Center/End]:

### Second Point

Draws an arc using three specified points on the arc's circumference. The first point is the start point (1). The third point is the endpoint (3). The second point (2) is a point on the circumference of the arc.

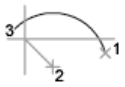
Specify end point of arc:*Specify a point (3)*


You can specify a three-point arc either clockwise or counterclockwise.

### Center

Specifies the center of the circle of which the arc is a part.

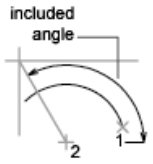
Specify center point of arc:
Specify end point of arc or [Angle/chord Length]:
**End Point** Using the center point (2), draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point through the third point (3).




The arc does not necessarily pass through this third point, as shown in the illustration.

**Angle** Draws an arc counterclockwise from the start point (1) using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

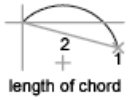
Specify included angle:*Specify an angle*




**Chord Length** Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.

Specify length of chord:*Specify a length*

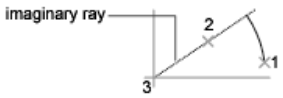



### End

Specifies the endpoint of the arc.

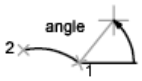
Specify end point of arc:
Specify center point of arc or [Angle/Direction/Radius]:

**Center Point** Draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point (3) through the second point specified (2).



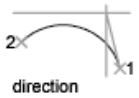

**Angle** Draws an arc counterclockwise from the start point (1) to an endpoint (2), with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle: *Enter an angle in degrees or specify an angle by moving the pointing device counterclockwise*



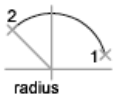

**Direction** Begins the arc tangent to a specified direction. It creates any arc, major or minor, clockwise or counterclockwise, beginning with the start point (1), and ending at an endpoint (2). The direction is determined from the start point.

Specify tangent direction for the start point of arc:




**Radius** Draws the minor arc counterclockwise from the start point (1) to the endpoint (2). If the radius is negative, the major arc is drawn.

Specify radius of arc:

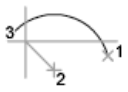



### Center

Specifies the center of the circle of which the arc is a part.

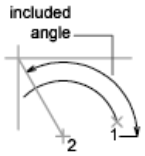
Specify center point of arc:
Specify start point of arc:
Specify end point of arc or [Angle/chord Length]:

**End Point** Draws an arc counterclockwise from the start point (2) to an endpoint that falls on an imaginary ray drawn from the center point (1) through a specified point (3).




**Angle** Draws an arc counterclockwise from the start point (2) using a center point (1) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

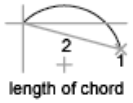
Specify included angle:




**Chord Length** Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.

Specify length of chord:




### Tangent to Last Line, Arc, or Polyline

Draws an arc tangent to the last line, arc, or polyline drawn when you press ENTER at the first prompt.


Specify end point of arc: *Specify a point (1)*

# ARCHIVE

## Quick Reference

Packages the current sheet set files to be archived

**archive**

The Archive a Sheet Set dialog box (page 87) is displayed.

If you enter **-archive** at the command prompt, options are displayed at the command prompt (page 94).

# Archive a Sheet Set Dialog Box

## Quick Reference

**archive**

Packages the files associated with the current sheet set so that they can be archived.

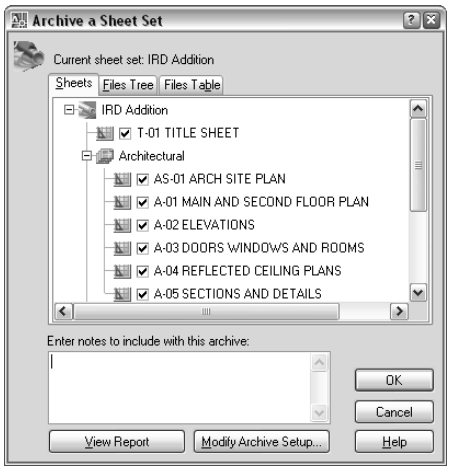
The Sheets tab lists the sheets to be included in the archive package in sheet order and according to subset organization. The Files Tree tab lists the files to be included in the package in a hierarchical tree format. The Files Table tab displays the files to be included in the package in a table format.

All files to be included in the archive package are indicated by a check mark next to the file name. To remove a file from the archive package, select the check box. Right-click in the file display area to display a shortcut menu, from which you can clear all check marks or apply check marks to all files.

**NOTE** Make sure that the files to be archived are not currently open in any application.

**Sheets Tab**

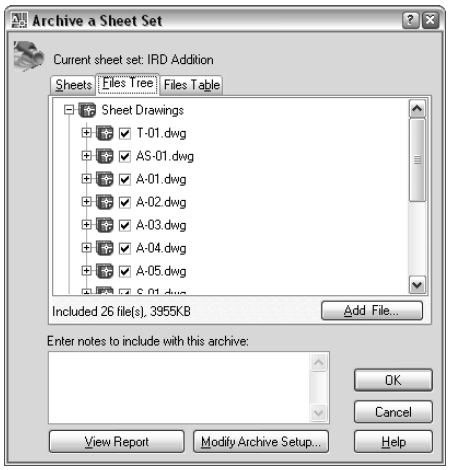
Lists the sheets to be included in the archive package in sheet order and according to subset organization. A sheet set must be open in the Sheet Set Manager. Archive can be selected from the shortcut menu displayed when the top-level sheet set node is right-clicked, or **archive** can be entered at the command prompt.




### Files Tree Tab

Lists the files to be included in the archive package in a hierarchical tree format. By default, all files associated with the current drawing (such as related xrefs, plot styles, and fonts) are listed. You can add files to the archive package or remove existing files. Related files that are referenced by URLs are not included in the archive package.

**Included** Reports the number of files included in the package and the size of the package.



### Files Table Tab

Displays the files to be included in the archive package in a table format. By default, all files associated with the current drawing (such as related xrefs, plot styles, and fonts) are listed. You can add files to the archive package or remove existing files. Related files that are referenced by URLs are not included in the archive package.

**Included** Reports the number of files included in the package and the size of the package.


### Add a File

Opens a standard file selection dialog box, in which you can select an additional file to include in the archive package. This button is available on both the Files Tree tab and the Files Table tab.

### Enter Notes to Be Included with This Archive

Provides a space where you can enter notes related to the archive package. The notes are included in the archive report. You can specify a template of default notes to be included with all your archive packages by creating an ASCII text file called *archive.txt.* This file must be saved to a location specified by the Support File Search Path option on the Files tab (page 948) in the Options dialog box.

### View Report

Displays report information that is included with the archive package. Includes any archive notes that you entered. If you have created a text file of default notes, the notes are also included in the report.

**Save As** Opens a File Save dialog box, in which you can specify a location in which to save a report file. Note that a report file is automatically included with all archive packages that you generate; by choosing Save As, you can save an additional copy of a report file for reference purposes.

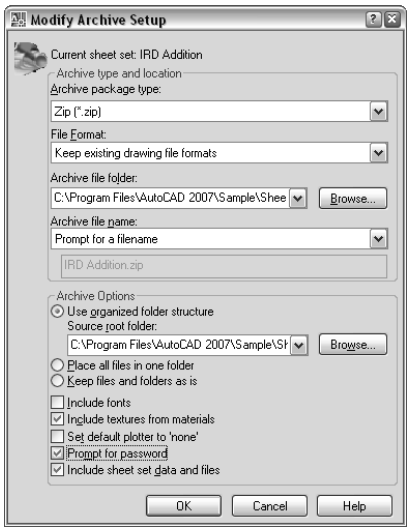
### Modify Archive Setup

Displays the Modify Archive Setup dialog box (page 90), in which you can specify options for the archive package.

# Modify Archive Setup Dialog Box

## Quick Reference

### archive



### Archive Package Type

Specifies the type of archive package created.

**Folder** Creates an archive package of uncompressed files in a new or existing folder.

**Self-Extracting Executable** Creates an archive package of files as a compressed, self-extracting executable file. Double-clicking the resulting EXE file decompresses the archive package and restores the files.

**Zip** Creates an archive package of files as a compressed ZIP file. To restore the files, you need a decompression utility such as the shareware application PKZIP or WinZip.

### File Format

Specifies the file format to which all drawings included in the archive package will be converted. You can select a drawing file format from the drop-down list.

### Archive File Folder

Specifies the location in which the archive package is created. Lists the last nine locations in which archive packages were created. To specify a new location, click Browse and navigate to the location you want.

If this field is left blank, the archive file is created in the folder containing the sheet set data (DST) file.

### Browse

Opens a standard file selection dialog box, in which you can navigate to a location where you create the archive package.

### Archive File Name

Specifies the method for naming the archive package. Displays the default file name for the archive package. This option is not available if the archive package type is set to Folder.

**Prompt for a File Name** Displays a standard file selection dialog box where you can enter the name of the archive package.

**Increment File Name if Necessary** Uses a logical default file name. If the file name already exists, a number is added to the end. This number is incremented each time a new archive package is saved.

**Overwrite if Necessary** Uses a logical default file name. If the file name already exists, the existing file is automatically overwritten.

### Archive Options

Provides options for the archive.

**Use Organized Folder Structure** Duplicates the folder structure for the files being transmitted. The root folder is the top-level folder within a hierarchical folder tree.

The following considerations apply:

- Relative paths remain unchanged. Relative paths outside the source root folder retain up to one level of the folder path above them, and are placed in the root folder.
- Absolute paths within the root folder tree are converted to relative paths. Absolute paths retain up to one level of the folder path above them, and are placed in the root folder.
- Absolute paths outside the root folder tree are converted to No Path and are moved to the root folder or to a folder within the root folder tree.
- A *Fonts* folder is created, if necessary.
- A *PlotCfgs* folder is created, if necessary.
- A *SheetSets* folder is created to hold all support files for sheet sets, if necessary. The sheet set data (DST) file, however, is placed in the root folder.

This option is not available if you're saving an archive package to an Internet location.

**Source Root Folder** Defines the source root folder for relative paths of drawing-dependent files, such as xrefs. This option determines the source root folder for the Use Organized Folder Structure option.

The source root folder also contains the sheet set data (DST) file when a sheet set is archived.

**Browse** Opens a standard file selection dialog box, in which you can navigate to specify a source root folder.

**Place All Files in One Folder** All files are installed to a single, specified target folder when the archive package is created.

**Keep Files and Folders As Is** Preserves the folder structure of all files in the archive package. This option is not available if you're saving an archive package to an Internet location.

**Include Fonts** Includes any associated font files (TXT and SHX) with the archive package.

**Include Textures from Materials** Includes textures with materials that are attached to objects or faces.

**Include Photometric Web Files** Includes photometric web files that are associated with web lights in the drawing.

**Set Default Plotter to 'None'** Changes the printer/plotter setting in the archive package to None.

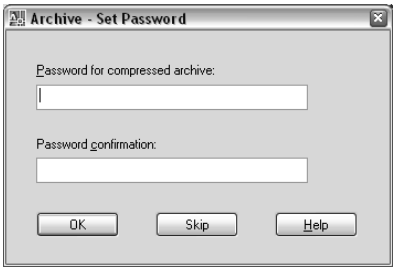
**Prompt for Password** Opens the Archive—Set Password dialog box (page 93), where you can specify a password for your archive package.

**Include Sheet Set Data and Files** Includes the sheet set data (DST) file, label block drawing files, callout block drawing files, and drawing template (DWT) files with the archive package.

# Archive - Set Password Dialog Box

## Quick Reference

**archive**



#### Password for Compressed Archive Package

Provides a space for an optional password for the archive package. When others attempt to open the archive package, they must provide this password to access the files. Password protection cannot be applied to folder archive packages.

#### Password Confirmation

Provides a space to confirm the password that you entered in the Password field. If the two passwords do not match, you are prompted to reenter them.

# -ARCHIVE

## Quick Reference

If you enter **-archive** at the command prompt, the following ARCHIVE command prompts are displayed.

Sheet Set name or [?] <*current*>: *Enter a predefined sheet set name, enter* **?** *to display a list of sheet sets, or press* ENTER

The sheet set name specifies a sheet set to use for the archive package. This option is available only when a sheet set is open.

Enter an option [Create archive package/Report only] <*Create*>: *Enter an option or press* ENTER
Password (press ENTER for none): *Enter a password or press* ENTER

You are only prompted for a password if you selected Prompt for Password in the archive setup.

# AREA

## Quick Reference

Calculates the area and perimeter of objects or of defined areas

Inquiry 

Tools ➤ Inquiry ➤ AreaAt the Command prompt, enter area.

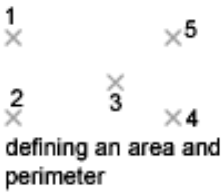
**area**

Specify first corner point (page 95) or [Object (page 95)/Add (page 96)/Subtract (page 97)]: *Specify a point (1) or enter an option*

### First Corner Point

Calculates the area and perimeter you define by specifying points. All points must lie in a plane parallel to the *XY* plane of the current user coordinate system (UCS).

Specify next corner point or press ENTER for total: *Specify a point (2)*



defining an area and perimeter


area defined

Continue to specify points to define a polygon and then press ENTER to complete the definition of the perimeter.

If you do not close the polygon, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated, that line length is added.

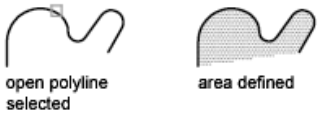
### Object

Calculates the area and perimeter of the selected object. You can calculate the area of circles, ellipses, splines, polylines, polygons, regions, and solids.

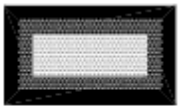
**NOTE** 2D solids (created with the *SOLID* command) do not have an area reported.

Select objects:

If you select an open polyline, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated that line length is ignored.



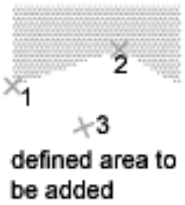

The centerline of a wide polyline is used to make area and perimeter (or length) calculations.




### Add

Turns on Add mode and keeps a running balance of the total area as you continue to define areas. The Add option calculates the individual areas and perimeters of defined areas and objects as well as the total area of all defined areas and objects. You can use the Subtract option to subtract specified areas from the total area.

Specify first corner point or [Object/Subtract]: *Specify a point (1) or enter an option*





**First Corner Point** Calculates the area and perimeter you define by selecting points. All points must lie in a plane parallel to the *XY* plane of the current UCS.

Specify next corner point or press ENTER for total (ADD mode): *Specify a point (2)*

Specify points to define a polygon (3). Press ENTER. AREA calculates the area and perimeter and returns the total area of all the areas defined by selecting points or objects since Add mode was turned on.

If you do not close the polygon, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated, that line length is added.

**Object** Calculates the area and perimeter of the selected object.

(ADD mode) Select objects:

AREA calculates the area and perimeter and returns the total area of all the areas defined by selecting points or objects since Add mode was turned on.

If you select an open polyline, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated that line length is ignored.

The centerline of a wide polyline is used to make area and perimeter calculations.

**Subtract** Turns on Subtract mode and keeps a running balance of the total area as you subtract specified areas.

#### Subtract

Similar to the Add option, but subtracts areas and perimeters.


# ARRAY

## Quick Reference

Creates multiple copies of objects in a pattern

Modify 

Modify ➤ ArrayAt the Command prompt, enter array.

**array**

The Array dialog box (page 98) is displayed. You can create rectangular or polar arrays by choosing the appropriate option. Each object in an array can be manipulated independently. If you select multiple objects, the objects are treated as one item to be copied and arrayed.

If you enter **-array** at the command prompt, options are displayed at the command prompt (page 103).

# Array Dialog Box

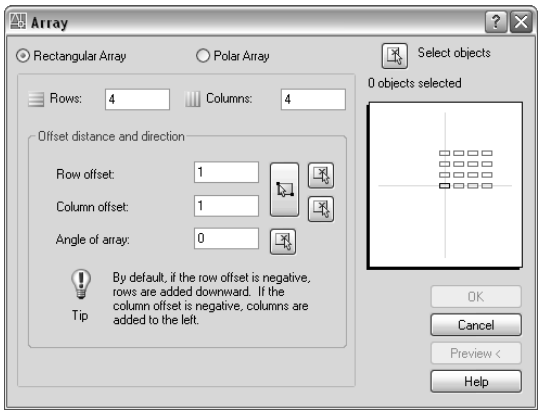
## Quick Reference

Modify 

Modify ➤ ArrayAt the Command prompt, enter array.

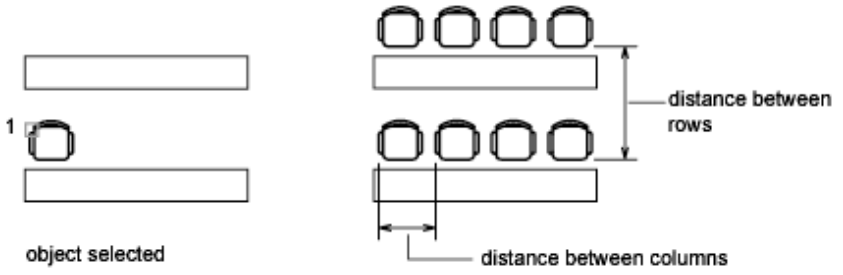
**array**

Creates multiple copies of objects in a pattern. Use the Rectangular Array (page 98) option to create an array of rows and columns of copies of the selected object. Use the Polar Array (page 100) option to create an array by copying the selected objects around a center point.




### Rectangular Array

Creates an array of rows and columns of copies of the selected object.




### Rows

Specifies the number of rows in the array.

If you specify one row, you must specify more than one column. If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. The limit is set by the MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter **(setenv "MaxArray" "200000")** at the Command prompt.

### Columns

Specifies the number of columns in the array.

If you specify one column, you must specify more than one row. If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. The limit is set by the MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter **(setenv "MaxArray" "200000")** at the Command prompt.

### Offset Distance and Direction

Provides a space for you to specify the distance and direction of the array's offset.

**Row Offset** Specifies the distance (in units) between rows. To add rows downward, specify a negative value. To specify row spacing with the pointing device, use the Pick Both Offsets button or the Pick Row Offset button.

**Column Offset** Specifies the distance (in units) between columns. To add columns to the left, specify a negative value. To specify column spacing with the pointing device, use the Pick Both Offsets button or the Pick Column Offset button.

**Angle of Array** Specifies the angle of rotation. This angle is normally 0, so the rows and columns are orthogonal with respect to the *X* and *Y* drawing axes of the current UCS. You can change the measurement conventions for angles using *UNITS*. The *ANGBASE* and *ANGDIR* system variables affect the angle of arrays.

**Pick Both Offsets** Temporarily closes the Array dialog box so that you can use the pointing device to set the row and column spacing by specifying two diagonal corners of a rectangle.

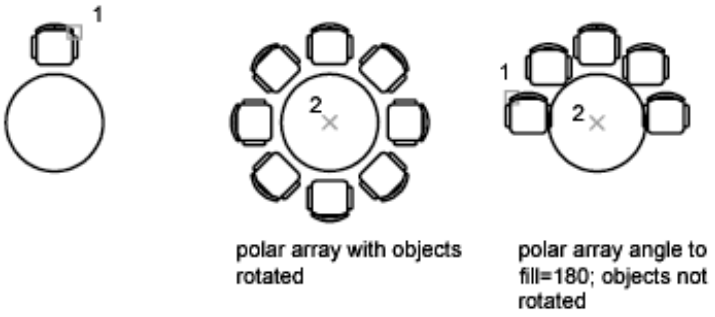
**Pick Row Offset** Temporarily closes the Array dialog box so that you can use the pointing device to specify the distance between rows. ARRAY prompts you to specify two points and uses the distance and direction between the points to specify the value in Row Offset.

**Pick Column Offset** Temporarily closes the Array dialog box so that you can use the pointing device to specify the distance between columns. ARRAY prompts you to specify two points and uses the distance and direction between the points to specify the value in Column Offset.

**Pick Angle of Array** Temporarily closes the Array dialog box so that you can specify the angle of rotation by entering a value or using the pointing device to specify two points. You can change the measurement conventions for angles using *UNITS*. The *ANGBASE* and *ANGDIR* system variables affect the angle of arrays.

### Polar Array

Creates an array by copying the selected objects around a specified center point.



### Center Point

Specifies the center point of the polar array. Enter coordinate values for *X* and *Y*, or choose Pick Center Point to use the pointing device to specify the location.

### Pick Center Point

Temporarily closes the Array dialog box so that you can use the pointing device to specify the center point in the drawing area.

### Method and Values

Specifies the method and values used to position objects in the polar array.

**Method** Sets the method used to position objects. This setting controls which of the Method and Value fields are available for specifying values. For example, if the method is Total Number of Items & Angle to Fill, the related fields are available for specifying values; the Angle Between Items field is not available.

**Total Number of Items** Sets the number of objects that appear in the resultant array. The default value is 4.

**Angle to Fill** Sets the size of the array by defining the included angle between the base points of the first and last elements in the array. A positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation. The default value is 360. A value of 0 is not permitted.

**Angle Between Items** Sets the included angle between the base points of the arrayed objects and the center of the array. Enter a positive value. The default direction value is 90.

---

**NOTE** You can choose the Pick buttons and use the pointing device to specify the values for Angle to Fill and Angle Between Items.

---

**Pick Angle to Fill** Temporarily closes the Array dialog box so that you can define the included angle between the base points of the first and last elements in the array. ARRAY prompts you to select a point relative to another point in the drawing area.

**Pick Angle Between Items** Temporarily closes the Array dialog box so that you can define the included angle between the base points of the arrayed objects and the center of the array. ARRAY prompts you to select a point relative to another point in the drawing area.

### Rotate Items as Copied

Rotates the items in the array, as shown in the preview area.

### More/Less

Turns the display of additional options in the Array dialog box on and off. When you choose More, additional options are displayed, and the name of this button changes to Less.

### Object Base Point

Specifies a new reference (base) point relative to the selected objects that will remain at a constant distance from the center point of the array as the objects are arrayed. To construct a polar array, ARRAY determines the distance from the array's center point to a reference (base) point on the last object selected. The point used depends on the type of object, as shown in the following table.

**Base point settings by object**

| Object type | Default base point |
|---|---|
| Arc, circle, ellipse | Center point |
| Polygon, rectangle | First corner |
| Donut, line, polyline, 3D polyline, ray, spline | Starting point |
| Block, paragraph text, single-line text | Insertion point |
| Construction lines | Midpoint |
| Region | Grip point |

**Set to Object's Default** Uses the default base point of the object to position the arrayed object. To manually set the base point, clear this option.

**Base Point** Sets a new *X* and *Y* base point coordinate. Choose Pick Base Point to temporarily close the dialog box and specify a point. After you specify a point, the Array dialog box is redisplayed.

**NOTE** To avoid unexpected results, set the base point manually if you are constructing a polar array and do not want to rotate the objects.

#### Select Objects

Specifies the objects used to construct the array. You can select objects before or after the Array dialog box is displayed. To select objects when the Array dialog box is displayed, choose Select Objects. The dialog box temporarily closes. When you finish selecting objects, press ENTER. The Array dialog box is redisplayed, and the number of objects selected is shown below the Select Objects button.

**NOTE** If you select multiple objects, the base point of the last selected object is used to construct the array.

#### Preview Area

Shows a preview image of the array based on the current settings in the dialog box. The preview image is dynamically updated when you move to another field after changing a setting.

#### Preview

Closes the Array dialog box and displays the array in the current drawing. Choose Modify to return to the Array dialog box to make changes.

# -ARRAY

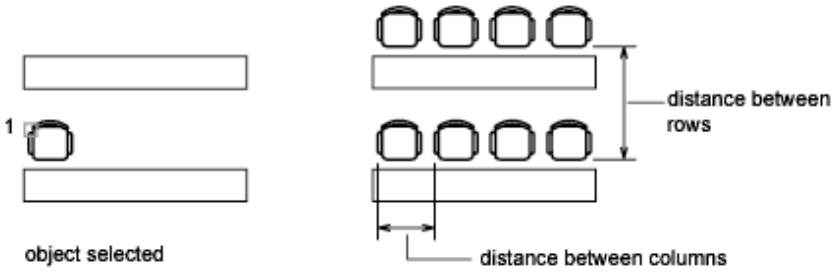
## Quick Reference

If you enter **-array** at the command prompt, the following ARRAY command prompts are displayed.

Select objects: *Use an object selection method*
Enter the type of array [Rectangular (page 104)/Polar (page 105)] <*current*>: *Enter an option or press* ENTER

### Rectangular

Creates an array of rows and columns of copies of the selected objects.




Enter the number of rows (---) <1>:*Enter a nonzero integer or press* ENTER
Enter the number of columns (|||) <1>:*Enter a nonzero integer or press* ENTER

If you specify one row, you must specify more than one column and vice versa.

The selected object, or cornerstone element, is assumed to be in the lower-left corner, and generates the array up and to the right.

The specified distance between the rows and columns includes the corresponding lengths of the object to be arrayed.

Enter the distance between rows or specify unit cell (---):

To add rows downward, specify a negative value for the distance between rows. ARRAY skips the next prompt if you specify two points for the opposite corners of a rectangle.

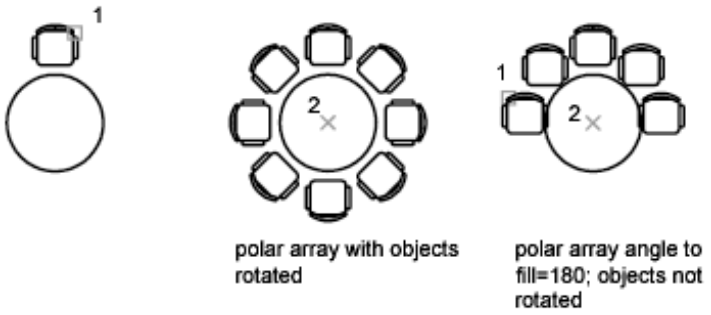
Specify the distance between columns (|||):

To add columns to the left, specify a negative value for the distance between columns. Rectangular arrays are constructed along a baseline defined by the current snap rotation. This angle is normally 0, so the rows and columns are orthogonal with respect to the *X* and *Y* drawing axes. The Rotate option of the *SNAP* command changes the angle and creates a rotated array. The *SNAPANG* system variable stores the snap rotation angle.

If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. The limit is set by the MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter **(setenv "MaxArray" "200000")** at the Command prompt.

### Polar

Creates an array by copying the selected objects around a specified center point.




Specify center point of array or [Base]:*Specify a point or enter* **b** *to specify a new base point*
**Center Point** Creates an array defined by a center point.

**Base** Specifies a new reference (base) point relative to the selected objects that will remain at a constant distance from the center point of the array as the objects are arrayed.

Specify the base point of objects: Specify a point

Enter the number of items in the array: *Enter a positive integer or press* ENTER

If you enter a value for the number of items, you must specify either the angle to fill or the angle between items. If you press ENTER (and do not provide the number of items), you must specify both.

Specify the angle to fill (+=ccw, -=cw) <360>: *Enter a positive integer for a counterclockwise rotation or a negative integer for a clockwise rotation*

You can enter **0** for the angle to fill only if you specify the number of items.

If you specify an angle to fill without providing the number of items, or if you specify the number of items and enter **0** as the angle to fill or press ENTER, the following prompt is displayed:

Angle between items: *Specify an angle*

If you specified the number of items and entered 0 as the angle to fill or pressed ENTER, ARRAY prompts for a positive or negative value to indicate the direction of the array:

Angle between items (+=ccw, -=cw): *Enter a positive integer for a counterclockwise rotation or a negative integer for a clockwise rotation*

ARRAY determines the distance from the array's center point to a reference point on the last object selected. The reference point used is the center point of a circle or arc, the insertion base point of a block or shape, the start point of text, and one endpoint of a line or trace.

Rotate arrayed objects? <Y>:*Enter* **y** *or* **n**, *or press* ENTER

In a polar array, the reference point of the last object in the selection set is used for all objects. If you defined the selection set by using window or crossing selection, the last object in the selection set is arbitrary. Removing an object from the selection set and adding it back forces that object to be the last object selected. You can also make the selection set into a block and replicate it.

# ARX

## Quick Reference

Loads, unloads, and provides information about ObjectARX applications

**arx**
Enter an option [? (page 106)/Load (page 106)/Unload (page 106)/Commands (page 107)/Options (page 107)]:

### ? - List Applications

Lists the currently loaded ObjectARX applications, which can be third-party programs or internal applications such as Render.

### Load

Displays the ObjectARX/DBX File dialog box (a standard file selection dialog box (page 931)). This option loads the specified ObjectARX application.

### Unload

Unloads the specified ObjectARX application.

Enter ARX/DBX file name to unload:

### Commands

Lists the AcEd-registered commands (AcEd-registered commands are described in the *ObjectARX Developer's Guide*).

### Options

Presents developer-related ObjectARX application options. These options are explained in greater detail in the *ObjectARX Developer's Guide*.

Enter an option [Group/CLasses/Services]: *Enter an option or press*ENTER
**Group** Causes the specified group of commands to be the first group searched when resolving the names of AutoCAD commands.

**Classes** Displays a class hierarchy of C++ classes derived from objects registered in the system.

**Services** Lists the names of all registered services.

# ATTACHURL

## Quick Reference

Attaches hyperlinks to objects or areas in a drawing

**attachurl**
Enter hyperlink insert option [Area (page 107)/Object (page 108)] <Object>: *Enter* **a** *or press* ENTER

### Area

Creates the URLLAYER layer, draws a polyline on that layer, and attaches a URL to the polyline.

First corner: *Click in the drawing to indicate the lower-left corner of the area*
Other corner: *Click to indicate the upper-right corner of the area*
Enter hyperlink <current drawing>: *Enter a URL*

The polyline that represents the area is displayed in the color assigned to URLLAYER. The default color is red. When you move the cursor over the area in the drawing, the cursor changes to a hyperlink cursor to indicate that a URL is attached to the area.

### Object

Attaches a URL to the selected object.

Select objects: *Use an object selection method, and press*ENTER*to end selection*
Enter hyperlink <current drawing>: *Enter a URL*

When you move the cursor over the object in the drawing, the cursor changes to a hyperlink cursor to indicate that a URL is attached to the object.

# ATTDEF

## Quick Reference

Creates an attribute definition

Draw ➤ Block ➤ Define AttributesAt the Command prompt, enter attdef.
**attdef**

The Attribute Definition dialog box (page 108) is displayed.

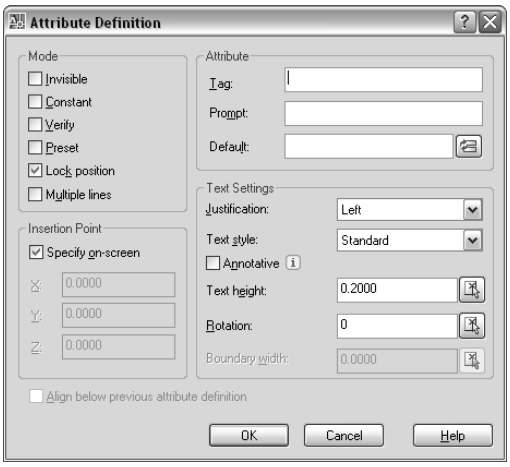
If you enter **-attdef** at the command prompt, options are displayed at the command prompt (page 111).

## Attribute Definition Dialog Box

### Quick Reference

Draw ➤ Block ➤ Define AttributesAt the Command prompt, enter attdef.
**attdef**

Defines the mode; attribute tag, prompt, and value; insertion point; and text settings for an attribute.




### Mode

Sets options for attribute values associated with a block when you insert the block in a drawing.

The default values are stored in the *AFLAGS* system variable. Changing the AFLAGS setting affects the default mode for new attribute definitions and does not affect existing attribute definitions.

**Invisible** Specifies that attribute values are not displayed or printed when you insert the block. *ATTDISP* overrides Invisible mode.

**Constant** Gives attributes a fixed value for block insertions.

**Verify** Prompts you to verify that the attribute value is correct when you insert the block.

**Preset** Sets the attribute to its default value when you insert a block containing a preset attribute.

**Lock Position** Locks the location of the attribute within the block reference. When unlocked, the attribute can be moved relative to the rest of the block using grip editing, and multiline attributes can be resized.

**Multiple Lines** Specifies that the attribute value can contain multiple lines of text. When this option is selected, you can specify a boundary width for the attribute.

---

**NOTE** In a dynamic block, an attribute's position must be locked for it to be included in an action's selection set.

---

### Attribute

Sets attribute data.

**Tag** Identifies each occurrence of an attribute in the drawing. Enter the attribute tag using any combination of characters except spaces. Lowercase letters are automatically changed to uppercase.

**Prompt** Specifies the prompt that is displayed when you insert a block containing this attribute definition. If you do not enter a prompt, the attribute tag is used as a prompt. If you select Constant in the Mode area, the Prompt option is not available.

**Default** Specifies the default attribute value.

**Insert Field Button** Displays the Field dialog box (page 578). You can insert a field as all or part of the value for an attribute.

**Multiline Editor Button** When Multiple Line mode is selected, displays an in-place text editor with a text formatting toolbar and ruler. Depending on the setting of the *ATTIPE* system variable, the Text Formatting toolbar displayed is either the abbreviated version, or the full version. For more information, see the In-Place Text Editor (page 873).

---

**NOTE** Several options in the full In-Place Text Editor are grayed out to preserve compatibility with single-line attributes.

---

### Insertion Point

Specifies the location for the attribute. Enter coordinate values or select Specify On-screen and use the pointing device to specify the placement of the attribute in relation to the objects that it will be associated with.

**Specify On-Screen** Displays a Start Point prompt when the dialog box closes. Use the pointing device to specify the placement of the attribute in relation to the objects that it will be associated with.

**X** Specifies the *X* coordinate of the attribute's insertion point.

**Y** Specifies the *Y* coordinate of the attribute's insertion point.

**Z** Specifies the *Z* coordinate of the attribute's insertion point.

### Text Settings

Sets the justification, style, height, and rotation of the attribute text.

**Justification** Specifies the justification of the attribute text. See *TEXT* for a description of the justification options.

**Text Style** Specifies a predefined text style for the attribute text. Currently loaded text styles are displayed. To load or create a text style, see *STYLE*.

**Annotative** Specifies that the attribute is . If the block is annotative, the attribute will match the orientation of the block. Click the information icon to learn more about annotative objects.

**Text Height** Specifies the height of the attribute text. Enter a value, or choose Height to specify a height with your pointing device. The height is measured from the origin to the location you specify. If you select a text style that has fixed height (anything other than 0.0), or if you select Align in the Justification list, the Height option is not available.

**Rotation** Specifies the rotation angle of the attribute text. Enter a value, or choose Rotation to specify a rotation angle with your pointing device. The rotation angle is measured from the origin to the location you specify. If you select Align or Fit in the Justification list, the Rotation option is not available.

**Boundary Width** Specifies the maximum length of the lines of text in a multiple-line attribute before wrapping to the next line. A value of 0.000 means that there is no restriction on the length of a line of text. Not available for single-line attributes.

#### Align Below Previous Attribute Definition

Places the attribute tag directly below the previously defined attribute. If you have not previously created an attribute definition, this option is not available.

# -ATTDEF

## Quick Reference

If you enter **-attdef** at the command prompt, the following ATTDEF command prompts are displayed.

Current attribute modes (page 112): Invisible=*current* Constant=*current* Verify=*current* Preset=*current* Lock position=*current* Annotative =*current* Multiple line =*current*
Enter an option to change [Invisible/Constant/Verify/Preset/Lock position/Annotative/Multiple lines] <done>:

Enter attribute tag name (page 113): *Enter any characters except spaces or exclamation points*
Enter attribute value (page 113): *Enter the appropriate text or press* ENTER *(this prompt is displayed only if you turned on Constant mode)*
Enter attribute prompt (page 113): *Enter the text for the prompt line or press* ENTER *(this prompt is not displayed if you turned on Constant mode)*
Enter default attribute value (page 113): *Enter the appropriate text or press* ENTER *(this prompt is not displayed if you turned on Constant mode)*
Specify location of multiline attribute (page 114): *Specify a point (this prompt is displayed only if you turned on Multiple line mode)*
Specify opposite corner (page 114): *Specify a point or enter an option (this prompt is displayed only if you turned on Multiple line mode)*

### Attribute Modes

The current value line indicates the current settings for each attribute mode (either Y for on or N for off). Entering **i**, **c**, **v**, **p**, **l**, **a**, or **m** toggles the modes on or off. Press ENTER when you have finished adjusting the mode settings. The *AFLAGS* system variable stores the current mode settings and can be used to set the default modes.

**Invisible** Specifies that attribute values are displayed when you insert the block. *ATTDISP* overrides Invisible mode.

**Constant** Gives attributes a fixed value for block insertions.

**Verify** Prompts for verification that the attribute value is correct when you insert the block.

**Preset** Sets the attribute to its default value when you insert a block containing a preset attribute.

**Lock Position** Locks the location of the attribute within the block reference. When unlocked, the attribute can be moved relative to the rest of the block using grip editing, and multiline attributes can be resized.

---

**NOTE** In a dynamic block, an attribute's position must be locked for it to be included in an action's selection set.

---

**Annotative** Specifies that the attribute is .

**Multiple Lines** Specifies that the attribute value can contain multiple lines of text. When this option is selected, you can specify a boundary width for the attribute.

### Attribute Tag Name

Specifies the attribute tag, which identifies each occurrence of an attribute in the drawing. The tag can contain any characters except spaces or exclamation marks (!). Lowercase letters are automatically changed to uppercase.

### Attribute Prompt

Specifies the prompt that is displayed when you insert a block containing this attribute definition. If you press ENTER, the attribute tag is used as the prompt. If you turn on Constant mode, this prompt is not displayed.

---

**NOTE** For single-line attributes, you can enter up to 256 characters. If you need leading blanks in the prompt or the default value, start the string with a backslash (\). To make the first character a backslash, start the string with two backslashes.

---

### Default Attribute Value

Specifies the default attribute value. The default attribute value appears when a block is inserted into your drawing. A default value is not required. If you turn on Constant mode, the Attribute Value prompt is displayed instead.

When Multiple Line mode is off, -ATTDEF then displays the same prompts as the TEXT command, using the attribute tag instead of requesting a text string.

Current text style: "Standard" Text height: 0.2000
Specify start point of text or [Justify / Style]: *Enter an option or press* ENTER
Specify paper text height <*current*>: *Specify a height, or press* ENTER

The Specify Paper Text Height prompt is displayed only if the current text style is .

For a description of each option, see *TEXT*.

---

**NOTE** For single-line attributes, you can enter up to 256 characters. If you need leading blanks in the prompt or the default value, start the string with a backslash (\). To make the first character a backslash, start the string with two backslashes.

---

When Multiple Line mode is on, -ATTDEF then displays several of the prompts used by the MTEXT command. For a description of each option, see MTEXT (page 872).

### Attribute Value (Constant Mode)

Specifies the value for a constant attribute. This prompt is displayed only if you turn on Constant mode.

ATTDEF then displays the same prompts as the TEXT command, using the attribute tag instead of requesting a text string.

Current text style: "Standard" Text height: 0.2000
Specify start point of text or [Justify / Style]: *Enter an option or press* ENTER

For a description of each option, see *TEXT*.

When Multiple Line mode is on, -ATTDEF then displays several of the prompts used by the MTEXT command. For a description of each option, see MTEXT (page 872).

#### Location of Multiline Attribute (Multiple Line Mode)

Specifies the first corner of the bounding box for the multiple-line text. This location is used as the starting point for the attribute.

#### Opposite Corner (Multiple Line Mode)

As you drag the pointing device to specify the opposite corner, a rectangle is displayed to show the location and width of the multiple-line text. The arrow within the rectangle indicates the direction of the text flow.

# ATTDISP

## Quick Reference

Globally controls visibility of block attributes in a drawing

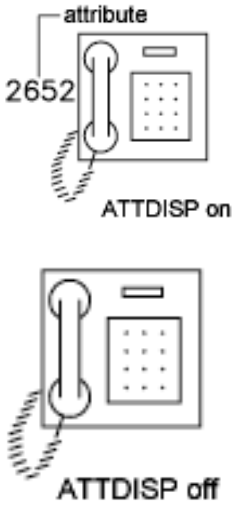
View ➤ Display ➤ Attribute DisplayDoes not exist in the menus.
**attdisp** (or '**attdisp** for transparent use)
Enter attribute visibility setting [Normal (page 114)/ON (page 114)/OFF (page 114)] <*current*>:

The drawing is regenerated after you change the visibility unless *REGENAUTO*, which controls automatic regeneration, is off. The current visibility of attributes is stored in the *ATTMODE* system variable.

**Normal** Retains the current visibility of each attribute. Visible attributes are displayed. Invisible attributes are not displayed.

**On** Makes all attributes visible.

**Off** Makes all attributes invisible.




# ATTEDIT

## Quick Reference

Changes attribute information in a block

Modify ➤ Object ➤ Attribute ➤ SingleAt the Command prompt, enter eattedit.
**attedit**
Select block reference: *Select a block with attributes*

The Edit Attributes dialog box (page 116) is displayed to edit attribute values for a specific block.

Modify ➤ Object ➤ Attribute ➤ GlobalAt the Command prompt, enter attedit.
**-attedit**

If you enter **-attedit** at the command prompt, options are displayed at the command prompt (page 117) to edit attribute values and properties independent of a block.

# Edit Attributes Dialog Box

## Quick Reference

Modify ➤ Object ➤ Attribute ➤ SingleAt the Command prompt, enter eattedit.
**attedit**

Changes attribute information in a block.

To change attribute properties such as position, height, and style, use -ATTEDIT.

**Block Name** Indicates the name of the selected block. The value of each attribute contained in the block is displayed in this dialog box.

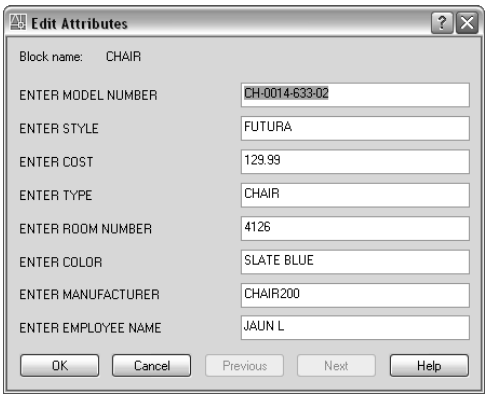
**List of Attributes** Displays the first eight attributes contained in the block. Edit the attribute values. If the block contains additional attributes, use Previous and Next to navigate through the list. You cannot edit attribute values on locked layers.

Multiple-line attributes display the In-Place Text Editor (page 873) with the Text Formatting toolbar and the ruler. Depending on the setting of the *ATTIPE* system variable, the Text Formatting toolbar displayed is either the abbreviated version, or the full version.

To use a field as the value, right-click and click Insert Field on the shortcut menu to display the Field dialog box (page 578).

**Previous** Displays the previous eight attribute values. Previous is available only if the selected block contains more than eight attributes and you used Next to display the additional attributes.

**Next** Displays the next eight attribute values. If the block contains no additional attributes, Next is unavailable.




# -ATTEDIT

## Quick Reference

If you enter **-attedit** at the command prompt, the following ATTEDIT command prompts are displayed.

Edit attributes one at a time? [Yes (page 117)/No (page 121)] <Y>: *Enter* **y** *or press* ENTER *to edit attributes one at a time, or enter* **n** *to edit attributes globally*

### Yes

Edits attributes one at a time. Attributes to be edited one at a time must be visible and parallel to the current UCS.

Enter block name specification <*>: *Press* ENTER*, or enter a block name or a partial block name with wild-card characters (? or *) to narrow the selection to specific blocks*
Enter attribute tag specification <*>: *Press* ENTER*, or enter a tag or a partial tag with wild-card characters (? or *) to narrow the selection to specific attributes*
Enter attribute value specification <*>: *Press* ENTER*, or specify a value or a value name with wild-card characters (? or *) to narrow the selection to specific attribute values*

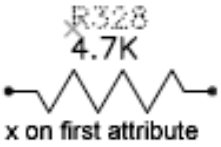
Attribute values are case sensitive.

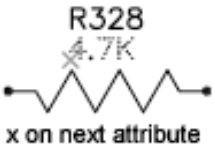
Select Attributes: *Select only attributes parallel to the current UCS*


The first attribute in the selection set is marked with an *X*. You can change any properties of the attribute you select.

Enter an option [Value/Position/Height/Angle/Style/Layer/Color/Next] <N>: *Enter the property to change, or press*ENTER*for the next attribute*

If the original attribute was defined with aligned or fit text, the prompt does not include Angle. The Height option is omitted for aligned text. For each of the options except Next, ATTEDIT prompts for a new value. The *X* remains on the current attribute until you move to the next attribute.




**Value**

Changes or replaces an attribute value.

Enter type of value modification [Change/Replace]: *Enter* **c** *or* **r** *or press* ENTER

**Change** Modifies a few characters of the attribute value.

Enter string to change: *Enter string to change or press* ENTER

Enter new string: *Enter replacement string or press* ENTER

Either string can be null. The ? and * characters are interpreted literally, not as wild-card characters.

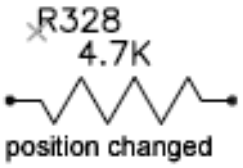
**Replace** Substitutes a new attribute value for the entire attribute value.

Enter new attribute value: *Enter a new attribute value or press* ENTER

If you press ENTER, the attribute value is empty (null).

### Position

Changes the text insertion point.

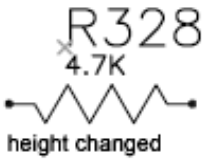



Specify new text insertion point: *Specify a point or press* ENTER

If the attribute is aligned, ATTEDIT prompts for both ends of a new text baseline.

### Height

Changes the text height.

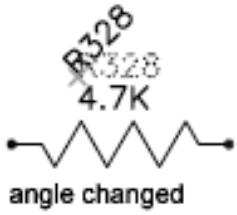



Specify new height <*current*>: *Enter a value, specify a point, or press* ENTER

When you specify a point, the height becomes the distance between the specified point and the start point of the text.

### Angle

Changes the rotation angle.

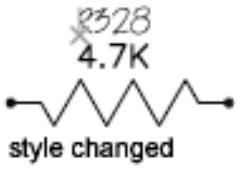



Specify new rotation angle <*current*>: *Enter a value, specify a point, or press* ENTER

If you specify a point, the text is rotated along an imaginary line between the specified point and the start point of the text.

### Style

Changes the style setting.




Enter new text style: *Enter a style name or press* ENTER

### Layer

Changes the layer.

Enter new layer name <*current*>: *Enter a layer name or press* ENTER

### Color

Changes the color.

You can enter a color from the AutoCAD Color Index (a color name or number), a true color, or a color from a color book.

Enter new color [Truecolor/COlorbook]<BYLAYER>: *Enter a color, enter* **t**, *enter* **co**, *or press* ENTER

You can enter a color name, a color number between 1 and 255, or **bylayer** or **byblock**.

**True Color** Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

**Color Book** Specifies a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* PANTONE®

If you enter a color book name, you are prompted to enter the color name in the color book.

Enter color name: *Enter the name of a color included in the selected color book, such as* PANTONE® 573

### Next

Moves to the next attribute in the selection set. If there are no more attributes, ATTEDIT ends.

### No

Edits more than one attribute at a time. Global editing applies to both visible and invisible attributes.

Editing attributes globally limits you to replacing a single text string with another text string. If you edit attributes one at a time, you can edit any or all of the attributes.

Performing global editing of attribute values.
Edit only attributes visible on screen? [Yes/No] <Y>: *Enter* **y** *or press* ENTER *to edit only visible attributes, or enter* **n** *to edit all attributes*
**Yes** Edits only visible attributes.

Enter block name specification <*>: *Press* ENTER*, or specify a block name or a partial block name with wild-card characters (? or *) to narrow the selection to specific blocks*

Enter attribute tag specification <*>: *Press* ENTER*, or specify a tag or a partial tag with wild-card characters (? or *) to narrow the selection to specific attributes*

Enter attribute value specification <*>: *Press* ENTER*, or specify a value or a partial value with wild-card characters (? or *) to narrow the selection to specific attribute values*

Attribute values are case sensitive. To select empty (null) attributes, which normally are not visible and cannot be selected, enter a backslash (\).

Select Attributes: *Select only attributes parallel to the current UCS*

Select the attribute you want to change.

Enter string to change: *Enter string to change or press* ENTER

Enter new string: *Enter replacement string or press* ENTER

Either string can be empty (null). The ? and * characters are interpreted literally, not as wild-card characters.

**No** Edits attributes whether they are visible or not. Changes to attributes are not reflected immediately. The drawing is regenerated at the end of the command unless *REGENAUTO*, which controls automatic regeneration, is off.

Enter block name specification <*>: *Enter a full block name or a partial block name with wild-card characters (? or *), or press* ENTER *to select attributes in all blocks*

Enter attribute tag specification <*>: *Enter a full attribute tag name or a partial name with wild-card characters (? or *), or press* ENTER *to select attributes in all attribute tags*

Enter attribute value specification <*>: *Enter a full attribute value or a partial value with wild-card characters (? or *), or press* ENTER *to select all attributes*

Attribute values are case sensitive. To select empty (null) attributes, which normally are not visible, enter a backslash (\).

The attributes that match the specified block name, attribute tag, and attribute value are selected.

Enter string to change: *Enter the attribute value you want to change, or press* ENTER *to append a new value to the existing value(s)*

Enter new string: *Enter the new value to replace the specified value or to append to selected values*

Either string can be empty (null). The ? and * characters are interpreted literally, not as wild-card characters.

# ATTEXT

## Quick Reference

Extracts attribute data, informational text associated with a block, into a file

**attext**

The Attribute Extraction dialog box (page 122) is displayed.

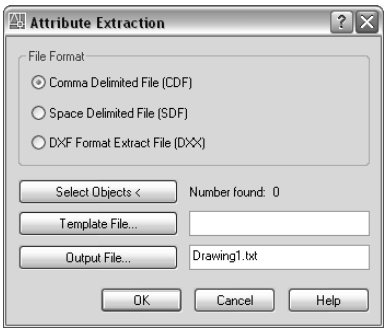
If you enter **-attext** at the command prompt, options are displayed at the command prompt (page 124).

# Attribute Extraction Dialog Box

## Quick Reference

**attext**

Specifies the file format for the attribute information, the objects from which you want to extract information, and the template and output file names for the information.




### File Format

Sets the format for the file into which you are extracting the attribute data.

**Comma Delimited File (CDF)** Generates a file containing one record for each block reference in the drawing that has at least one matching attribute tag in the template file. Commas separate the fields of each record. Single quotation marks enclose the character fields.

**Space Delimited File (SDF)** Generates a file containing one record for each block reference in the drawing that has at least one matching attribute tag in the template file. The fields of each record have a fixed width; therefore, field separators or character string delimiters are not appropriate.

**DXF Format Extract File (DXX)** Produces a subset of the AutoCAD Drawing Interchange File format containing only block reference, attribute, and end-of-sequence objects. DXF™ format extraction requires no template. The file name extension *.dxx* distinguishes the output file from normal DXF files.

### Select Objects

Closes the dialog box so you can use the pointing device to select blocks with attributes. When the Attribute Extraction dialog box reopens, Number Found shows the number of objects you selected.

### Number Found

Indicates the number of objects you selected using Select Objects.

### Template File

Specifies a template extraction file for CDF and SDF formats. Enter the file name in the box, or choose Template File to search for existing template files using a standard file selection dialog box (page 931). The default file extension is *.txt*. If you select DXF under File Format, the Template File option is not available.

For information about creating a template file, see Attach Data to Blocks (Block Attributes) in the *User's Guide*.

### Output File

Specifies the file name and location for the extracted attribute data. Enter the path and file name for the extracted attribute data, or choose Output File to search for existing template files using a standard file selection dialog box (page 931). The *.txt* file name extension is appended for CDF or SDF files and the *.dxx* file name extension for DXF files.

# -ATTEXT

## Quick Reference

If you enter **-attext** at the command prompt, the following ATTEXT command prompts are displayed.

Enter extraction type or enable object selection [Cdf (page 124)/Sdf (page 124)/Dxf (page 125)/Objects (page 125)] <C>: *Enter an option or press*ENTER
**CDF: Comma-Delimited File** Generates a file containing one record for each block reference in the drawing. Commas separate the fields of each record. Single quotation marks enclose the character fields.

In the Select Template File dialog box, enter the name of an existing attribute extraction template file.

In the Create Extract File dialog box, enter the name for the output file. The extract file's file name extension is *.txt* for CDF or SDF format.

**SDF: Space-Delimited File** Generates a file containing one record for each block reference in the drawing. The fields of each record have a fixed width; therefore, field separators or character string delimiters are not used.

In the Select Template File dialog box, enter the name of an existing attribute extraction template file.

In the Create Extract File dialog box, enter the name for the output file. The extract file's file name extension is *.txt* for CDF or SDF format.

**DXF: Drawing Interchange File** Produces a subset of the AutoCAD Drawing Interchange File format containing only block reference, attribute, and end-of-sequence objects. DXF-format extraction requires no template. The file name extension *.dxx* distinguishes the output file from normal DXF files.

In the Create Extract File dialog box, enter the name for the output file. The extract file's file name extension is *.dxx* for DXF format.

**Objects** Selects objects whose attributes you want to extract.

Select object: *Use an object selection method*

Enter attribute extraction type [Cdf/Sdf/Dxf] <C>: *Enter an option or press*ENTER

# ATTIPEDIT

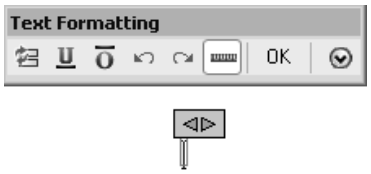
## Quick Reference

Changes the textual content of an attribute within a block

**attipedit**
Select attribute to edit: *Select an attribute within a block*

If you select a single-line attribute, displays the In-Place Text Editor (page 873) without the Text Formatting toolbar and the ruler. Right-click to display options.

If you select a multiple-line attribute, displays the In-Place Text Editor (page 873) with the Text Formatting toolbar and the ruler. Depending on the setting of the *ATTIPE* system variable, the Text Formatting toolbar displayed is either the abbreviated version shown, or the full version.




Use the abbreviated version for compatibility with previous AutoCAD releases and editing operations. Use the full version for additional text formatting options.

**NOTE** Not all MTEXT formatting options are available for multiline attributes even with the full In-Place Editor.

#### Mask the Objects Behind Attribute Text

If the text in a multiple-line attribute overlaps other objects within the block, you can hide those objects that interfere with the text with a background mask. Click the Options button in the Text Formatting toolbar and click Background Mask to display the Background Mask dialog box (page 883).

# ATTREDEF

## Quick Reference

Redefines a block and updates associated attributes

**attredef**
Enter the name of the block you wish to redefine:
Select objects for new block:
Select objects:
Insertion base point of new block: *Specify a point*

New attributes assigned to existing block references use their default values. Old attributes in the new block definition retain their old values. Any old attributes that are not included in the new block definition are deleted.

**WARNING** ATTREDEF removes any format or property changes made with the ATTEDIT or EATTEDIT commands. It also deletes any extended data associated with the block, and might affect dynamic blocks and blocks created by third-party applications.

# ATTSYNC

## Quick Reference

Updates all instances of a specified block with the current attributes defined for the block

Modify II 
**attsync**
Enter an option [?/Name/Select] <Select>:

You are prompted for the names of blocks you want to update with the current attributes defined for the blocks.

Use this command to update all instances of a block containing attributes that was redefined using the BLOCK or BEDIT commands. ATTSYNC does not change any values assigned to attributes in existing blocks.

---

**NOTE** Use the ATTREDEF command to redefine and update blocks in one command.

---

Entering **?** displays a list of all block definitions in the drawing. Enter the name of the block you want to update.

Pressing ENTER allows you to use your pointing device to select the block whose attributes you want to update.

If a block you specify does not contain attributes or does not exist, an error message is displayed, and you are prompted to specify another block.

---

**WARNING** ATTSYNC removes any format or property changes made with the ATTEDIT or EATTEDIT commands. It also deletes any extended data associated with the block, and might affect dynamic blocks and blocks created by third-party applications.

---

# AUDIT

## Quick Reference

Evaluates the integrity of a drawing and corrects some errors

File ➤ Drawing Utilities ➤ AuditAt the Command prompt, enter audit.
**audit**
Fix any errors detected? [Yes/No] <N>: *Enter* **y** *or* **n***, or press* ENTER

For easy access, AUDIT places all objects for which it reports errors in the Previous selection set. However, editing commands affect only the objects that belong to the current paper space or model space.

If you set the *AUDITCTL* system variable to 1, AUDIT creates an ASCII file describing problems and the action taken and places this report in the same directory as the current drawing, with the file extension *.adt*.

If a drawing contains errors that AUDIT cannot fix, use *RECOVER* to retrieve the drawing and correct its errors.

# AUTOPUBLISH

## Quick Reference

Publishes drawings to DWF files automatically

**autopublish**
AutoPublish DWF or specify override [Location] <AutoPublish>:*Press* ENTER *or enter* **l**
**Location** Specify a directory the published drawings from the Select a Folder for Generated Files dialog box (a standard file selection dialog box (page 931)).

To view information on the published drawings, click the Plotting Details Report Available icon in the tray on the right side of the status bar. Clicking this icon opens the Plot and Publish Details dialog box (page 1516), which provides information about your completed plot and publish jobs. This information is also saved to the Plot and Publish log file. The shortcut menu for this icon also provides an option to view the most recently published Design Web Format™(DWF™).

# Auto Publish Options Dialog Box

## Quick Reference

Specifies options for publishing automatically.

### Auto-Publish Options

Specifies the output folder location where Design Web Format (DWF) files are saved when you publish drawing sheets.

**Publish On** Specifies when the publishing will take place.

**Location** Specifies where DWF files are saved when you publish drawings.

**Include** Specifies whether the Model, layouts or bothe the model and layouts are saved when you publish drawings.

### General DWF Options

Specifies options for creating a single-sheet DWF file.

**DWF Type** Specifies that a single-sheet DWF file or a single multisheet DWF file is generated for all the sheets listed in the Publish dialog box (page 1124).

**Password Protection** Specifies options for protecting DWF files with passwords.

### DWF Data Options

Lists and allows you to specify the data that you can optionally include in the DWF file.

**Layer Information** Specifies whether layer information is included in the published DWF file.

**NOTE** Layer information for 3D DWF entries does not get published.

**Block Information** Specifies whether block property and attribute information is included in the published DWF files.

**NOTE** You must set block information to Include in order for block template file information to be available.

**Block Template File** Allows you to create a new block template (BLK) file, edit an existing block template file, or use the settings of a previously created block template file.

Create opens the Publish Block Template dialog box (page 1131), in which you can create a new block template.

Edit opens the Select Block Template dialog box (a standard file selection dialog box (page 931)), in which you can select an existing block template to modify.

# B Commands

# 3

**In this chapter**

- BOX
- BPARAMETER
- BREAK
- BREP
- BROWSER
- BSAVE
- BSAVEAS
- BVHIDE
- BVSHOW
- BVSTATE

# BACKGROUND

## Quick Reference

Obsolete. BACKGROUND functionality has been merged into the VIEW command.

**view**

To add a background to a drawing, create a named view, assign a background to the view, and then restore the view to see the background.

Refer to the *VIEW* command.

# BACTION

## Quick Reference

Adds an action to a dynamic block definition

Block Editor 
**bedit ➤ baction**

You can only use the BACTION command in the Block Editor (page 160). Actions define how the geometry of a dynamic block reference will move or change when the custom properties of a block reference are manipulated in a drawing. You associate actions with parameters.

Select parameter: *Select a parameter in the block definition with which to associate the action*
Select action type [Array (page 134)/Move (page 134)/Scale (page 135)/Stretch (page 136)/Polar Stretch (page 137)]: *Select an action type to associate with the parameter (the available actions types depend on the type of parameter selected)*

If you select a rotation parameter, a rotate (page 138) action is automatically added to the dynamic block definition and associated with the selected parameter.

If you select a flip parameter, a flip (page 139) action is automatically added to the dynamic block definition and associated with the selected parameter.

If you select a lookup parameter, a lookup (page 139) action is automatically added to the dynamic block definition and associated with the selected parameter.

### Array

Adds an array action to the current dynamic block definition. An array action can be associated with a linear, polar, or XY parameter. Specifies that the selection set of objects will array when the action is triggered in a dynamic block reference.

Specify selection set for action
Select objects: *Select objects for the action's selection set*

If you selected a linear or polar parameter, the following prompts are displayed:

Enter the distance between columns (|||): *Enter a value for the distance between the columns of arrayed objects*
Specify action location: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Enter the distance between rows or specify unit cell (---): *Enter a value for the distance between the rows or enter two values separated by a comma for each of the two points for a unit cell for the arrayed objects*
Enter the distance between columns (|||): *Enter a value for the distance between the columns of arrayed objects*
Specify action location: *Specify a location in the block definition for the action*

### Move

Adds a move action to the current dynamic block definition. A move action can be associated with a point, linear, polar, or XY parameter. Specifies that the selection set of objects will move when the action is triggered in a dynamic block reference.

If you selected a point parameter, the following prompts are displayed:

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected a linear or polar parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset/XY]: *Specify a location in the block definition for the action*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases by a specified number the angle of the associated parameter when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

**XY** Specifies whether the distance that is applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Scale

Adds a scale action to the current dynamic block definition. A scale action can only be associated with a linear, polar, or XY parameter. Specifies that the selection set of objects will scale relative to the base point defined by the scale action when the action is triggered in a dynamic block reference.

If you selected a linear or polar parameter, the following prompts are displayed:

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Base type]: *Specify a location in the block definition for the action or enter* **base type** *to specify the type of base point*

If you selected an XY parameter, the following prompts are displayed:

Select parameter: *Select a parameter in the block definition to associate with the action*

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Base type/XY]: *Specify a location in the block definition for the action, enter* **base type** *to specify the type of base point, or enter* **xy** *to specify the distance type*

### Base Type

Specifies the type of base point to use for the action.

Enter base point type [Dependent/Independent] <Dependent>: *Enter the type of base point for the action or press* ENTER *to use a dependent base point*
**Dependent** Specifies that the dynamic block reference is scaled relative to the base point of the parameter with which the scale action is associated. An offset can be specified for the base point, which is maintained relative to the parameter.

Specify base point location <0,0>: *Enter two values separated by a comma or press* ENTER *to select 0,0*

**Independent** Specifies that the dynamic block reference is scaled relative to an independent base point you specify. An independent base point is shown in the Block Editor as an X marker.

Specify base point location <*XY location of associated parameter's base point*>: *Enter two values separated by a comma or press* ENTER *to select the location of the associated parameter's base point*

### XY

Specifies whether the distance applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Stretch

Adds a stretch action to the current dynamic block definition. A stretch action can be associated with a point, linear, polar, or XY parameter. Specifies that the selection set of objects will stretch or move when the action is triggered in a dynamic block reference.

If you selected a linear or polar parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects: *Select objects to add to or remove from the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point/Xcorner/Ycorner] <Xcorner>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the X corner*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects: *Select objects to add to or remove from the action's selection set*
Specify action location or [Multiplier/Offset/XY]: *Specify a location in the block definition for the action.*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases the angle of the associated parameter, by a specified number, when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

**XY** Specifies whether the distance that is applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Polar Stretch

Adds a polar stretch action to the current dynamic block definition. A polar stretch action can only be associated with a polar parameter. Specifies that the selection set of objects will stretch or move when the action is triggered in a dynamic block reference.

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects: *Select objects to add to the action's selection set for the stretch action*
Specify objects to rotate only
Select objects: *Select objects that will rotate only and not stretch*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases the angle of the associated parameter, by a specified number, when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

### Rotate

Adds a rotate action to the current dynamic block definition. A rotate action can only be associated with a rotation parameter. Specifies that the selection set of objects will rotate when the action is triggered in a dynamic block reference.

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Base type]: *Specify a location in the block definition for the action or enter* **base type** *to specify the type of base point*

### Base Type

Specifies the type of base point to use for the action.

Enter base point type [Dependent/Independent] <Dependent>: *Enter the type of base point for the action or press* ENTER *to use a dependent base point*
**Dependent** Specifies that the dynamic block reference rotates about the associated parameter's base point.

Specify base point location <current>: *Enter two values separated by a comma or press* ENTER *to select the current location of the parameter's base point*

**Independent** Specifies that the dynamic block reference rotates about a specified base point.

Specify base point location <*XY location of associated parameter's base point*>: *Enter two values separated by a comma or press* ENTER *to select the location of the associated parameter's base point*

#### Flip

Adds a flip action to the current dynamic block definition. A flip action can only be associated with a flip parameter. Specifies that the selection set of objects will flip around the reflection line of the flip parameter when the action is triggered in a dynamic block reference.

Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location: *Specify a location in the block definition for the action*

#### Lookup

Adds a lookup action to the current dynamic block definition. A lookup action can only be associated with a lookup parameter. When you add a lookup action to the block definition, the Property Lookup Table dialog box (page 176) is displayed

Specify action location: *Specify a location in the block definition for the action*

# BACTIONSET

## Quick Reference

Specifies the selection set of objects associated with an action in a dynamic block definition

Select an action in the block definition. Right-click in the Block Editor drawing area. Click Action Selection Set, and then click an option.
**bedit ➤ bactionset**

Re-specifies the selection set of objects associated with an action by creating a new selection set or by adding to or removing objects from the existing selection set. You can only use the BACTIONSET command in the Block Editor (page 160).

Select action: *Select an action in the current dynamic block definition*

Specify selection set for action object [New (page 140)/Modify (page 140)] <New>:
*Enter* **modify** *to modify the existing selection set or press* ENTER *to create a new selection set*

### New

If you selected a move, scale, rotate, flip, or array action, the following prompt is displayed:

Specify selection set for action
Select objects: *Select objects for the new selection set and then press* ENTER

If you selected a stretch action, the following prompts are displayed:

Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Select objects to stretch
Select objects: *Select objects for the new selection set and then press* ENTER

If you selected a polar stretch action, the following prompts are displayed:

Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects: *Select objects to stretch*
Specify objects to rotate only
Select objects: *Select objects that will rotate only and not stretch*

### Modify

If you selected a move, scale, rotate, flip, or array action, the following prompt is displayed:

Select objects to add to action set or [Remove]: *Select objects to add to or remove from the action's selection set that will rotate only and not stretch*

If you selected a stretch action, the following prompts are displayed:

Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Select objects to stretch
Select objects to add to action set or [Remove]: *Select objects to add to or remove from the action's selection set*

If you selected a polar stretch action, the following prompts are displayed:

Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects to add to action set or [Remove]: *Select objects to add to or remove from the action's selection set*
Specify objects to rotate only
Select objects to add to action set or [Remove]: *Select objects to add to or remove from the action's selection set that will rotate only and not stretch*

# BACTIONTOOL

## Quick Reference

Adds an action to a dynamic block definition

**bedit ➤ bactiontool**

The BACTIONTOOL command is used in the Block Editor (page 160) by action tools on the Actions tab of the Block Authoring Palettes. Actions define how the geometry of a dynamic block reference will move or change when the custom properties of a block reference are manipulated in a drawing. You associate actions with parameters.

Select action type [Array (page 141)/Lookup (page 142)/Flip (page 142)/Move (page 142)/Rotate (page 143)/Scale (page 144)/sTretch (page 145)/Polar stretch (page 146)]:

### Array

Adds an array action to the current dynamic block definition. An array action can be associated with a linear, polar, or XY parameter. Specifies that the selection set of objects will array when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify selection set for action
Select objects: *Select objects for the action's selection set*

If you selected a linear or polar parameter, the following prompts are displayed:

Enter the distance between columns (|||): *Enter a value for the distance between the columns of arrayed objects*

Specify action location: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Enter the distance between rows or specify unit cell (---): *Enter a value for the distance between the rows or enter two values separated by a comma for each of the two points for a unit cell for the arrayed objects*
Enter the distance between columns (|||): *Enter a value for the distance between the columns of arrayed objects*
Specify action location: *Specify a location in the block definition for the action*

### Lookup

Adds a lookup action to the current dynamic block definition. A lookup action can only be associated with a lookup parameter. When you add a lookup action to the block definition, the Property Lookup Table dialog box (page 176) is displayed

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify action location: *Specify a location in the block definition for the action*

### Flip

Adds a flip action to the current dynamic block definition. A flip action can only be associated with a flip parameter. Specifies that the selection set of objects will flip when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location: *Specify a location in the block definition for the action*

### Move

Adds a move action to the current dynamic block definition. A move action can be associated with a point, linear, polar, or XY parameter. Specifies that the selection set of objects will move when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*

If you selected a point parameter, the following prompts are displayed:

Specify selection set for action

Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected a linear or polar parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Multiplier/Offset/XY]: *Specify a location in the block definition for the action*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases the angle of the associated parameter, by a specified number, when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

**XY** Specifies whether the distance applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Rotate

Adds a rotate action to the current dynamic block definition. A rotate action can only be associated with a rotation parameter. Specifies that the selection set of objects will rotate when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify selection set for action

Select objects: *Select objects for the action's selection set*
Specify action location or [Base type]: *Specify a location in the block definition for the action or enter* **base type** *to specify the type of base point*

### Base Type

Specifies the type of base point to use for the action.

Enter base point type [Dependent/Independent] <Dependent>: *Enter the type of base point for the action or press* ENTER *to use a dependent base point*
**Dependent** Specifies that the dynamic block reference rotates about the associated parameter's base point.

Specify base point location <current>: *Enter two values separated by a comma or press* ENTER *to select the current location of the parameter's base point*

**Independent** Specifies that the dynamic block reference rotates about a specified base point.

Specify base point location <*XY location of associated parameter's base point*>: *Enter two values separated by a comma or press* ENTER *to select the location of the associated parameter's base point*

### Scale

Adds a scale action to the current dynamic block definition. A scale action can only be associated with a linear, polar, or XY parameter. Specifies that the selection set of objects will scale when the action is triggered in a dynamic block reference.

If you selected a linear or polar parameter, the following prompts are displayed:

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Base type]: *Specify a location in the block definition for the action or enter* **base type** *to specify the type of base point*

If you selected an XY parameter, the following prompts are displayed:

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify selection set for action
Select objects: *Select objects for the action's selection set*
Specify action location or [Base type/XY]: *Specify a location in the block definition for the action, enter* **base type** *to specify the type of base point, or enter* **xy** *to specify the distance type*

### Base Type

Specifies the type of base point to use for the action.

Enter base point type [Dependent/Independent] <Dependent>: *Enter the type of base point for the action or press* ENTER *to use a dependent base point*
**Dependent** Specifies that the dynamic block reference is scaled relative to the base point of the parameter with which the scale action is associated. An offset can be specified for the base point, which is maintained relative to the parameter.

Specify base point location <0,0>: *Enter two values separated by a comma or press* ENTER *to select 0,0*

**Independent** Specifies that the dynamic block reference is scaled relative to an independent base point you specify. An independent base point is shown in the Block Editor as an X marker.

Specify base point location <*XY location of associated parameter's base point*>: *Enter two values separated by a comma or press* ENTER *to select the location of the associated parameter's base point*

### XY

Specifies whether the distance that is applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Stretch

Adds a stretch action to the current dynamic block definition. A stretch action can be associated with a point, linear, polar, or XY parameter. Specifies that the selection set of objects will stretch or move when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*

If you selected a linear or polar parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*

Specify objects to stretch
Select objects: *Select objects to add to or remove from the action's selection set*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*

If you selected an XY parameter, the following prompts are displayed:

Specify parameter point to associate with action or enter [sTart point/Second point/Xcorner/Ycorner] <Xcorner>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the X corner*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*
Specify objects to stretch
Select objects: *Select objects to add to or remove from the action's selection set*
Specify action location or [Multiplier/Offset/XY]: *Specify a location in the block definition for the action.*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases the angle of the associated parameter, by a specified number, when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

**XY** Specifies whether the distance applied to the action is the XY parameter's *X* distance, *Y* distance, or *XY* distance from the parameter's base point.

Enter XY distance type [X/Y/XY] <XY>: *Enter the distance type or press* ENTER *to select XY*

### Polar Stretch

Adds a polar stretch action to the current dynamic block definition. A polar stretch action can only be associated with a polar parameter. Specifies that the selection set of objects will stretch or move when the action is triggered in a dynamic block reference.

Select parameter: *Select a parameter in the block definition to associate with the action*
Specify parameter point to associate with action or enter [sTart point/Second point] <Second>: *Specify a parameter point to associate with the action or press* ENTER *to associate the action with the second point*
Specify first corner of stretch frame or [CPolygon]: *Specify the first corner of the stretch frame or enter* **cpolygon** *to create a polygon of any shape as the stretch frame*
Specify opposite corner: *Specify the opposite corner of the stretch frame*

Specify objects to stretch
Select objects: *Select objects to add to or remove from the action's selection set*
Specify objects to rotate only
Select objects: *Select objects that will rotate only and not stretch*
Specify action location or [Multiplier/Offset]: *Specify a location in the block definition for the action*
**Multiplier** Changes the associated parameter value by a specified factor when the action is triggered.

Enter distance multiplier <1.0000>: *Enter a value or press* ENTER *to select 1.0000*

**Offset** Increases or decreases the angle of the associated parameter, by a specified number, when the action is triggered.

Enter angle offset <0>: *Enter a value or press* ENTER *to select 0*

# BASE

## Quick Reference

Sets the insertion base point for the current drawing

Draw ➤ Block ➤ BaseAt the Command prompt, enter base.
**base** (or '**base** for transparent use)
Enter base point <*current*>: *Specify a point or press* ENTER

The base point is expressed as coordinates in the current UCS. When you insert or externally reference the current drawing into other drawings, this base point is used as the insertion base point.

# BASSOCIATE

## Quick Reference

Associates an action with a parameter in a dynamic block definition

**bedit ➤ bassociate**

Associates an *orphaned* action with a parameter. You can only use the BASSOCIATE command in the Block Editor (page 160). An action becomes orphaned when the parameter with which it is associated is removed from the block definition.

Select action object: *Select an action in the current block definition that is not associated with a parameter*
Select parameter to associate with action: *Select a parameter to associate with the action (if you selected a lookup action, you can select one or more lookup parameters)*

If you selected an action and parameter combination that requires that the action be associated with a key point on the parameter, prompts are displayed to select the parameter point to associate with the action.

# BATTMAN

## Quick Reference

Edits attribute properties of a block definition

Modify II 
Modify ➤ Object ➤ Attribute ➤ Block Attribute ManagerAt the Command prompt, enter battman.
**battman**

The Block Attribute Manager (page 148) is displayed.

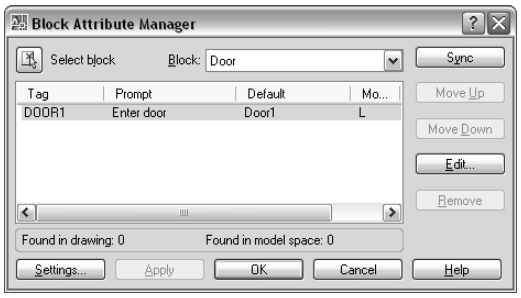
If the current drawing does not contain any blocks with attributes, a message is displayed.

# Block Attribute Manager

## Quick Reference

Modify II
Modify ➤ Object ➤ Attribute ➤ Block Attribute ManagerAt the Command prompt, enter battman.
**battman**

Manages the attribute definitions for blocks in the current drawing. You can edit the attribute definitions in blocks, remove attributes from blocks, and change the order in which you are prompted for attribute values when inserting a block.




Attributes of the selected block are displayed in the attribute list. By default, Tag, Prompt, Default, Mode, and Annotative attribute properties are displayed in the attribute list. You can specify which attribute properties you want displayed in the list by choosing Settings.

For each selected block, a description below the attribute list identifies the number of its instances in the current drawing and in the current layout.

**Select Block** Allows you to use your pointing device to select a block from the drawing area. When you choose Select Block, the dialog box closes until you select a block from the drawing or cancel by pressing ESC.

If you modify attributes of a block and then select a new block before you save the attribute changes you made, you are prompted to save the changes before selecting another block.

**Block** Lists all block definitions in the current drawing that have attributes. Select the block whose attributes you want to modify.

**List of Attributes** Displays the properties of each attribute in the selected block.

**Blocks Found in Drawing** The number of instances of the selected block in the current drawing.

**Blocks Found in Current Space** The number of instances of the selected block in the current model space or layout.

**Sync** Updates all instances of the selected block with the attribute properties currently defined. This does not affect any values assigned to attributes in each block.

**Move Up** Moves the selected attribute tag earlier in the prompt sequence. The Move Up button is not available when a constant attribute is selected.

**Move Down** Moves the selected attribute tag later in the prompt sequence. The Move Down button is not available when a constant attribute is selected.

**Edit** Opens the Edit Attribute dialog box (page 150), where you can modify attribute properties.

**Remove** Removes the selected attribute from the block definition. If Apply Changes to Existing References is selected in the Settings dialog box before you choose Remove, the attribute is removed from all instances of the block in the current drawing. The Remove button is not available for blocks with only one attribute.

**Settings** Opens the Block Attribute Settings dialog box (page 153), where you can customize how attribute information is listed in the Block Attribute Manager.

**Apply** Applies the changes you made, but leaves the dialog box open.

# Edit Attribute Dialog Box

## Quick Reference

Modify II
Modify ➤ Object ➤ Attribute ➤ Block Attribute ManagerAt the Command prompt, enter battman.
**battman**

Allows you to edit attributes for a block definition.

**Block Name** Displays the name of the block whose attributes are to be edited.

**Auto Preview Changes** Controls whether or not the drawing area is immediately updated to display any visible attribute changes you make. If Auto Preview Changes is selected, changes are immediately visible. If Auto Preview Changes is cleared, changes are not immediately visible.

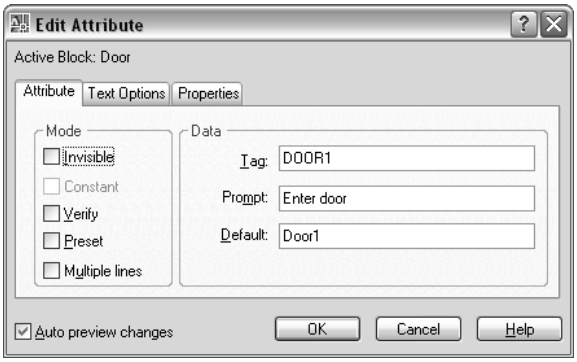
Clearing Auto Preview Changes results in a small improvement in performance. Auto Preview Changes is not available if Apply Changes to Existing References is not selected.

The Edit Attribute dialog box contains the following tabs:

- Attribute (page 151)
- Text Options (page 152)
- Properties (page 153)

### Attribute Tab (Edit Attribute Dialog Box)

Defines how a value is assigned to an attribute and whether or not the assigned value is visible in the drawing area, and sets the string that prompts users to enter a value. The Attribute tab also displays the tag name that identifies the attribute.




#### Mode

Mode options determine whether and how attribute text appears.

**Invisible** Displays or hides the attribute in the drawing area. If selected, hides the attribute value in the drawing area. If cleared, displays the attribute value.

**Constant** Identifies whether the attribute is set to its default value. You cannot change this property. If a check mark is shown in the check box, the attribute is set to its default value and cannot be changed. If the check box is empty, you can assign a value to the attribute.

**Verify** Turns value verification on and off. If selected, prompts you to verify the values you assign to the attribute when inserting a new instance of the block. If this option is cleared, verification is not performed.

**Preset** Turns default value assignment on and off. If selected, sets the attribute to its default value when the block is inserted. If cleared, ignores the attribute's default value and prompts you to enter a value when inserting the block.

**Multiple Lines** Indicates whether the attribute was defined as a Multiple Lines attribute and can contain multiple lines of text.

#### Data

Data options set the attribute text that is displayed.

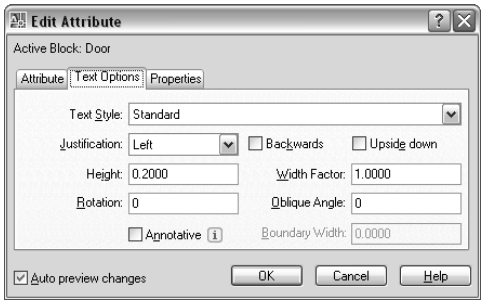
**Tag** Sets the identifier assigned to the attribute.

**Prompt** Sets the text for the prompt that is displayed when you insert the block.

**Default** Sets the default value assigned to the attribute when you insert the block.

### Text Options Tab (Edit Attribute Dialog Box)

Sets the properties that define the way an attribute's text is displayed in the drawing. Change the color of attribute text on the Properties tab.




**Text Style** Specifies the text style for attribute text. Default values for this text style are assigned to the text properties displayed in this dialog box.

**Justification** Specifies how attribute text is justified.

**Height** Specifies the height of the attribute text.

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

**Annotative** Specifies that the attribute is . Click the information icon to learn more about annotative objects.

**Backwards** Specifies whether or not the text is displayed backwards.

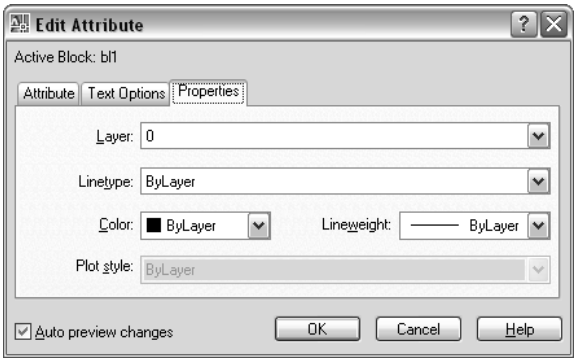
**Upside Down** Specifies whether or not the text is displayed upside down.

**Width Factor** Sets the character spacing for attribute text. Entering a value less than **1.0** condenses the text. Entering a value greater than **1.0** expands it.

**Oblique Angle** Specifies the angle that attribute text is slanted away from its vertical axis.

### Properties Tab (Edit Attribute Dialog Box)

Defines the layer that the attribute is on and the color, lineweight, and linetype for the attribute's line. If the drawing uses plot styles, you can assign a plot style to the attribute using the Properties tab.




**Layer** Specifies the layer that the attribute is on.

**Linetype** Specifies the linetype of attribute text.

**Color** Specifies the attribute's text color.

**Plot Style** Specifies the plot style of the attribute.

If the current drawing uses color-dependent plot styles, the Plot Style list is not available.

**Lineweight** Specifies the lineweight of attribute text.

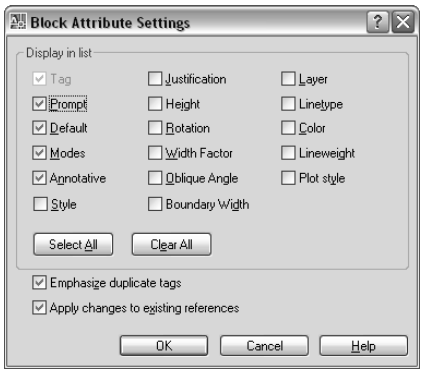
Changes you make to this option are not displayed if the LWDISPLAY system variable is off.

## Block Attribute Settings Dialog Box

### Quick Reference

Modify II
Modify ➤ Object ➤ Attribute ➤ Block Attribute ManagerAt the Command prompt, enter battman.
**battman**

Controls the appearance of the attribute list in the Block Attribute Manager.




**Display in List** Specifies the properties to be displayed in the attribute list. Only the selected properties are displayed in the list. The Tag property is always selected.

**Select All** Selects all properties.

**Clear All** Clears all properties.

**Emphasize Duplicate Tags** Turns duplicate tag emphasis on and off. If this option is selected, duplicate attribute tags are displayed in red type in the attribute list. If this option is cleared, duplicate tags are not emphasized in the attribute list.

**Apply Changes to Existing References** Specifies whether or not to update all existing instances of the block whose attributes you are modifying. If selected, updates all instances of the block with the new attribute definitions. If cleared, updates only new instances of the block with the new attribute definitions.

You can choose Sync in the Block Attribute Manager to apply changes immediately to existing block instances. This temporarily overrides the Apply Changes to Existing References option.

# BATTORDER

## Quick Reference

Specifies the order of attributes for a block

bedit ➤ **battorder**

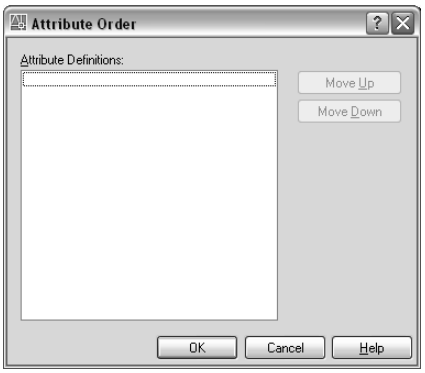
Displays the Attribute Order dialog box (page 155), which controls the order in which attributes are listed and prompted for when you insert or edit a block reference. You can only use the BATTORDER command in the Block Editor (page 160).

## Attribute Order Dialog Box

### Quick Reference

bedit ➤ **battorder**

Specifies the order in which attributes are listed and prompted for when you insert or edit a block reference.




**Attribute Definitions** Lists the attribute definitions in the current block.

**Move Up** Moves the selected attribute definition up in the list.

**Move Down** Moves the selected attribute definition down in the list.

# BAUTHORPALETTE

### Quick Reference

Opens the Block Authoring Palettes window in the Block Editor.

Block Editor 
bedit ➤ bauthorpalette

Displays the Block Authoring Palettes window in the Block Editor (page 160). You can open the Block Authoring Palettes window only from the Block Editor. If you enter **bauthorpalette** at the command prompt while the Block Editor is closed, the following message is displayed, "BAUTHORPALETTE command only allowed in Block Editor."

The Block Authoring Palettes window contains the following tabs:

- Parameters tab
- Actions tab
- Parameter Sets tab

# BAUTHORPALETTECLOSE

## Quick Reference

Closes the Block Authoring Palettes window in the Block Editor

Block Editor 
bedit ➤ bauthorpaletteclose

You can close the Block Authoring Palettes window only from the Block Editor (page 160). If you enter **bauthorpaletteclose** at the command prompt while you are not in the Block Editor, the following message is displayed, "BAUTHORPALETTECLOSE command only allowed in Block Editor."

# BCLOSE

## Quick Reference

Closes the Block Editor

In the Block Editor, right-click in the drawing area. Click Close Block Editor

**bedit ➤ bclose**

Closes the Block Editor (page 160). If you have modified the block definition since it was last saved, you are prompted to save or discard the changes.

# BCYCLEORDER

## Quick Reference

Changes the cycling order of grips for a dynamic block reference

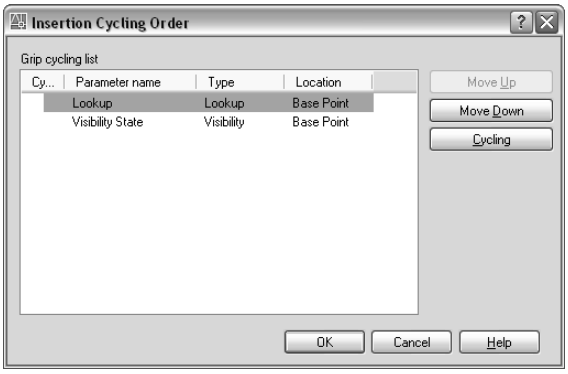
**bedit ➤ bcycleorder**

Displays the Insertion Cycling Order dialog box (page 157). You can only use the BCYCLORDER command in the Block Editor (page 160).

# Insertion Cycling Order Dialog Box

## Quick Reference

Specifies the grip-cycling order for the insertion point of a dynamic block reference. When you insert a dynamic block reference in a drawing, you can use the CTRL key to cycle through the grips that have cycling turned on in the block definition. The grip you select as you cycle is used as the insertion point for the block reference.




**Grip Cycling List** Lists the grips in the dynamic block definition. A check mark in the cycling column indicates that cycling is turned on for that grip.

**Move Up** Moves the selected grip up in the cycling order for the dynamic block reference.

**Move Down** Moves the selected grip down in the cycling order for the dynamic block reference.

**Cycling** Turns cycling on or off for the selected grip. A check mark in the cycling column indicates that cycling is turned on for that grip.

# BEDIT

## Quick Reference

Opens the Edit Block Definition dialog box and then the Block Editor

Standard 
Tools ➤ Block EditorAt the Command prompt, enter bedit.
Select a block reference. Right-click in the drawing area. Click Block Editor.
**bedit**

The Edit Block Definition dialog box (page 158) is displayed. Select a block definition to edit or entering a name for a new block definition to create, then click OK to open the Block Editor (page 160).

When the *BLOCKEDITLOCK* system variable is set to 1, the Block Editor cannot be opened.

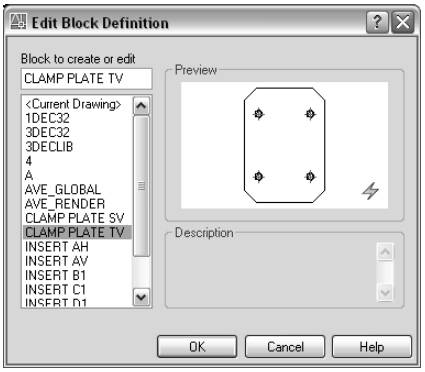
## Edit Block Definition Dialog Box

### Quick Reference

Standard 
Tools ➤ Block EditorAt the Command prompt, enter bedit.
**bedit**

In the Edit Block Definition dialog box, you can select from a list of block definitions that are saved in the drawing to edit in the Block Editor. You can also enter a name for a new block definition to create in the Block Editor.

When you click OK, the Edit Block Definition dialog box closes, and the Block Editor is displayed. If you selected a block definition from the list in the Edit Block Definition dialog box, that block definition is displayed and is available for editing in the Block Editor. If you entered a name for a new block definition, the Block Editor is displayed, and you can start adding objects to the block definition.




**Name** Specifies the name of the block to edit or create in the Block Editor. If you select <Current Drawing>, the current drawing is opened in the Block Editor. After you add dynamic elements to the drawing, you can then save it and insert it as a dynamic block reference in a drawing.

**Name List (Unlabeled)** Displays a list of block definitions that are saved in the current drawing. When you select a block definition from the list, the name is displayed in the Name box. When you click OK, this block definition is opened in the Block Editor. When you select <Current Drawing>, the current drawing is opened in the Block Editor.

**Preview** Displays a preview of the selected block definition. A lightning bolt icon indicates that the block is a dynamic block.

**Description** Displays the block definition description specified in the Block area of the Properties palette in the Block Editor.

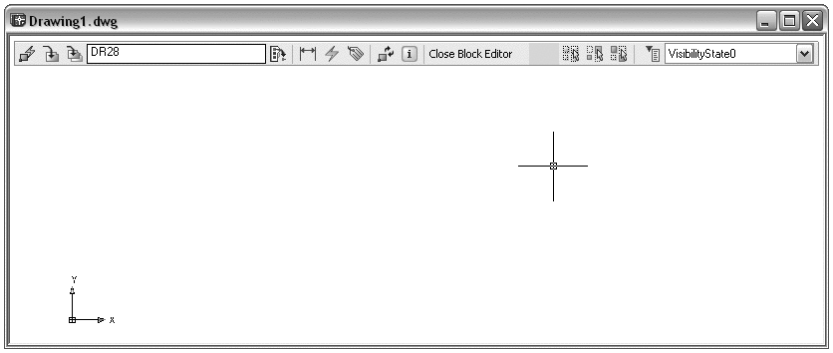
**OK** Opens the selected or new block definition in the Block Editor.

# Block Editor

## Quick Reference

Standard 
Select a block reference. Right-click in the drawing area. Click Block Editor.
Tools ➤ Block EditorAt the Command prompt, enter bedit.
**bedit**

You use the Block Editor to define dynamic behavior for a block definition. In the Block Editor, you add parameters and actions, which define custom properties and dynamic behavior. The Block Editor contains a special authoring area in which you can draw and edit geometry as you would in the drawing area.




The following commands are used for creating dynamic blocks and are available only in the Block Editor:

- *BACTION*
- *BACTIONSET*
- *BACTIONTOOL*
- *BASSOCIATE*
- *BATTORDER*
- *BAUTHORPALETTE*
- *BAUTHORPALETTECLOSE*

- *BCLOSE*
- *BCYCLEORDER*
- *BGRIPSET*
- *BLOOKUPTABLE*
- *BPARAMETER*
- *BSAVE*
- *BSAVEAS*
- *BVHIDE*
- *BVSHOW*
- *BVSTATE*

When the *BLOCKEDITLOCK* system variable is set to 1, the Block Editor cannot be opened.

The Block Editor also provides a Block Editor toolbar (page 161) and Block Authoring palettes that contain tools for creating dynamic blocks. The Block Authoring Palettes window contains the following tabs:

- Parameters tab (page 162)
- Actions tab (page 164)
- Parameter Sets tab (page 165)

### Block Editor Toolbar

Provides tools for working in the Block Editor, creating dynamic blocks, and working with visibility states.


**Edit or Create Block Definition** Displays the Edit Block Definition dialog box (page 158).

**Save Block Definition** Saves the current Block Definition.

**Save Block As** Displays the Save Block As dialog box (page 202), in which you can save a copy of the current block definition under a new name.

**Name** Displays the name of the current block definition.

**Authoring Palettes** Displays or hides the Block Authoring palettes, which provide tools for adding parameters and actions to the dynamic block definition.

**Parameter** Runs the *BPARAMETER* command, which adds a parameter to the dynamic block definition.

**Action** Runs the *BACTION* command, which adds an action to the dynamic block definition.

**Define Attribute** Displays the Attribute Definition dialog box (page 108), in which you can define the mode, attribute tag, prompt, value, insertion point, and text options for an attribute.

**Update Parameter and Action Text Size** Runs the *REGEN* command, which regenerates the display in the Block Editor and updates the text, arrowhead, icon, and grips sizes of parameters and actions. When you zoom in and out in the Block Editor, the text, arrowhead, icon, and grip size changes relative to the zoom factor. When you regenerate the display in the Block Editor, text, arrowheads, icons, and grips display at their specified values.

**Learn About Dynamic Blocks** Displays demonstrations in the New Features Workshop for creating dynamic blocks.

**Close Block Editor** Closes the Block Editor and prompts you to save or discard any changes to the current block definition.

**Visibility Mode** Sets the *BVMODE* system variable, which dims or hides objects made invisible for the current visibility state.

**Make Visible** Runs the *BVSHOW* command, so that you can make objects visible for the current visibility state or for all visibility states.

**Make Invisible** Runs the *BVHIDE* command, so that you can make objects invisible for the current visibility state or for all visibility states.

**Manage Visibility States** Displays the Visibility States dialog box (page 205), in which you can create, delete, rename, and make current visibility states.

**Visibility State** Specifies the current visibility state displayed in the Block Editor.

### Parameters Tab (Block Authoring Palettes Window)

Provides tools for adding parameters to a dynamic block definition in the Block Editor. Parameters specify positions, distances, and angles for geometry in the block reference. When you add a parameter to a dynamic block definition, it defines one or more custom properties for the block.

**Point Parameter** Runs the *BPARAMETER* command, Point parameter option, which adds a point parameter to the dynamic block definition and defines custom *X* and *Y* properties for the block reference. A point parameter defines an X and Y location in the drawing. In the Block Editor, a point parameter looks similar to an ordinate dimension.

**Linear Parameter** Runs the *BPARAMETER* command, Linear parameter option, which adds a linear parameter to the dynamic block definition and defines a custom distance property for the block reference. A linear parameter shows the distance between two anchor points. A linear parameter constrains grip movement along a preset angle. In the Block Editor, a linear parameter looks similar to an aligned dimension.

**Polar Parameter** Runs the *BPARAMETER* command, Polar parameter option, which adds a polar parameter to the dynamic block definition and defines custom distance and angle properties for the block reference. A polar parameter shows the distance between two anchor points and displays an angle value. You can use both grips and the Properties palette to change both the distance value and the angle. In the Block Editor, a polar parameter looks similar to an aligned dimension.

**XY Parameter** Runs the *BPARAMETER* command, XY parameter option, which adds an XY parameter to the dynamic block definition and defines custom horizontal and vertical distance properties for the block reference. An XY parameter shows the X and Y distances from the base point of the parameter. In the Block Editor, an XY parameter displays as a pair of dimensions (horizontal and vertical). These dimensions share a common base point.

**Rotation Parameter** Runs the *BPARAMETER* command, Rotation parameter option, which adds a rotation parameter to the dynamic block definition and defines a custom angle property for the block reference. A rotation parameter defines an angle. In the Block Editor, a rotation parameter displays as a circle.

**Alignment Parameter** Runs the *BPARAMETER* command, Alignment parameter option, which adds an alignment parameter to the dynamic block definition. An alignment parameter defines an X and Y location and an angle. An alignment parameter always applies to the entire block and needs no action associated with it. An alignment parameter allows the block reference to automatically rotate around a point to align with other objects in the drawing. An alignment parameter affects the angle property of the block reference. In the Block Editor, an alignment parameter looks like an alignment line.

**Flip Parameter** Runs the *BPARAMETER* command, Flip parameter option, which adds a flip parameter to the dynamic block definition and defines a custom flip property for the block reference. A flip parameter flips objects. In the Block Editor, a flip parameter displays as a reflection line. Objects can be

flipped about this reflection line. A flip parameter displays a value that shows if the block reference has been flipped or not.

**Visibility Parameter** Runs the *BPARAMETER* command, Visibility parameter option, which adds a visibility parameter to the dynamic block definition and defines a custom visibility property for the block reference. A visibility parameter allows you to create visibility states and to control the visibility of objects in the block. A visibility parameter always applies to the entire block and needs no action associated with it. In a drawing, you click the grip to display a list of visibility states available for the block reference. In the Block Editor, a visibility parameter displays as text with an associated grip.

**Lookup Parameter** Runs the *BPARAMETER* command, Lookup parameter option, which adds a lookup parameter to the dynamic block definition and defines custom lookup properties for the block reference. A lookup parameter defines a custom property that you can specify or set to evaluate to a value from a list or table you define. It can be associated with a single lookup grip. In the block reference, you click the grip to display a list of available values. In the Block Editor, a lookup parameter displays as text.

**Base Point Parameter** Runs the *BPARAMETER* command, Base Point parameter option, which adds a base point parameter to the dynamic block definition. A base point parameter defines a base point for the dynamic block reference relative to the geometry in the block. A base point parameter cannot be associated with any actions, but can belong to an action's selection set. In the Block Editor, a base point parameter displays as a circle with crosshairs

### Actions Tab (Block Authoring Palettes Window)

Provides tools for adding actions to a dynamic block definition in the Block Editor. Actions define how the geometry of a dynamic block reference will move or change when the custom properties of a block reference are manipulated in a drawing. You associate actions with parameters.

**Move Action** Runs the *BACTIONTOOL* command, Move action option, which adds a move action to the dynamic block definition when you associate the action with a point, linear, polar, or XY parameter. A move action is similar to the *MOVE* command. In a dynamic block reference, a move action causes objects to move a specified distance and angle.

**Scale Action** Runs the *BACTIONTOOL* command, Scale action option, which adds a scale action to the dynamic block definition when you associate the action with a linear, polar, or XY parameter. A scale action is similar to the *SCALE* command. In a dynamic block reference, a scale action causes its

selection set to scale when the associated parameter is edited by moving grips or by using the Properties palette.

**Stretch Action** Runs the *BACTIONTOOL* command, Stretch action option, which adds a stretch action to the dynamic block definition when you associate the action with a point, linear, polar, or XY parameter. A stretch action causes objects to move and stretch a specified distance in a specified location.

**Polar Stretch Action** Runs the *BACTIONTOOL* command, Polar Stretch action option, which adds a polar stretch action to the dynamic block definition when you associate the action with a polar parameter. A polar stretch action rotates, moves, and stretches objects a specified angle and distance when the key point on the associated polar parameter is changed through a grip or the Properties palette

**Rotate Action** Runs the *BACTIONTOOL* command, Rotate action option, which adds a rotate action to the dynamic block definition when you associate the action with a rotation parameter. A rotate action is similar to the *ROTATE* command. In a dynamic block reference, a rotate action causes its associated objects to rotate when the associated parameter is edited through a grip or the Properties palette.

**Flip Action** Runs the *BACTIONTOOL* command, Flip action option, which adds a flip action to the dynamic block definition when you associate the action with a flip parameter. With a flip action you can flip a dynamic block reference about a specified axis called a reflection line.

**Array Action** Runs the *BACTIONTOOL* command, Array action, which adds an array action to the dynamic block definition when you associate the action with a linear, polar, or XY parameter. An array action causes its associated objects to copy and array in a rectangular pattern when the associated parameter is edited through a grip or the Properties palette.

**Lookup Action** Runs the *BACTIONTOOL* command, Lookup action option, which adds a lookup action to the dynamic block definition. When you add a lookup action to a dynamic block definition and associate it with a lookup parameter, it creates a lookup table. You can use a lookup table to assign custom properties and values to a dynamic block.

### Parameter Sets Tab (Block Authoring Palettes Window)

Provides tools for adding a parameter and at least one action at the same time to a dynamic block definition in the Block Editor. When you add a parameter set to a dynamic block, the actions are automatically associated with the parameter. After you add a parameter set to a dynamic block, you double-click

the yellow alert icon (or use the *BACTIONSET* command) and follow the command prompts to associate the action with a selection set of geometry.

**Point Move** Runs the *BPARAMETER* command, Point parameter option with one grip specified, which adds a point parameter to the dynamic block definition. Automatically adds a move action associated with the point parameter.

**Linear Move** Runs the *BPARAMETER* command, Linear parameter option with one grip specified, which adds a linear parameter to the dynamic block definition. Automatically adds a move action associated with the endpoint of the linear parameter.

**Linear Stretch** Runs the *BPARAMETER* command, Linear parameter option with one grip specified, which adds a linear parameter to the dynamic block definition. Automatically adds a stretch action associated with the linear parameter.

**Linear Array** Runs the *BPARAMETER* command, Linear parameter option with one grip specified, which adds a linear parameter to the dynamic block definition. Automatically adds an array action associated with the linear parameter.

**Linear Move Pair** Runs the *BPARAMETER* command, Linear parameter option with two grips specified, which adds a linear parameter to the dynamic block definition. Automatically adds a two move action, one associated with the base point and one associated with the endpoint of the linear parameter.

**Linear Stretch Pair** Runs the *BPARAMETER* command, Linear parameter option with two grips specified, which adds a linear parameter to the dynamic block definition. Automatically adds a two stretch actions, one associated with the base point and one associated with the endpoint of the linear parameter.

**Polar Move** Runs the *BPARAMETER* command, Polar parameter option with one grip specified, which adds a polar parameter to the dynamic block definition. Automatically adds a move action associated with the polar parameter.

**Polar Stretch** Runs the *BPARAMETER* command, Polar parameter option with one grip specified, which adds a polar parameter to the dynamic block definition. Automatically adds a stretch action associated with the polar parameter.

**Polar Array** Runs the *BPARAMETER* command, Polar parameter option with one grip specified, which adds a polar parameter to the dynamic block definition. Automatically adds an array action associated with the polar parameter.

**Polar Move Pair** Runs the *BPARAMETER* command, Polar parameter option with two grips specified, which adds a polar parameter to the dynamic block definition. Automatically adds two move actions, one associated with the base point and one associated with the endpoint of the polar parameter.

**Polar Stretch Pair** Runs the *BPARAMETER* command, Polar parameter option with two grips specified, which adds a polar parameter to the dynamic block definition. Automatically adds two stretch actions, one associated with the base point and one associated with the endpoint of the polar parameter.

**XY Move** Runs the *BPARAMETER* command, XY parameter option with one grip specified, which adds an XY parameter to the dynamic block definition. Automatically adds a move action associated with the endpoint of the XY parameter.

**XY Move Pair** Runs the *BPARAMETER* command, XY parameter option with two grips specified, which adds an XY parameter to the dynamic block definition. Automatically adds two move actions, one associated with the base point and one associated with the endpoint of the XY parameter.

**XY Move Box Set** Runs the *BPARAMETER* command, XY parameter option with four grips specified, which adds an XY parameter to the dynamic block definition. Automatically adds four move actions, one associated with each key point on the XY parameter.

**XY Stretch Box Set** Runs the *BPARAMETER* command, XY parameter option with four grips specified, which adds an XY parameter to the dynamic block definition. Automatically adds four stretch actions, one associated with each key point on the XY parameter.

**XY Array Box Set** Runs the *BPARAMETER* command, XY parameter option with four grips specified, which adds an XY parameter to the dynamic block definition. Automatically adds an array action associated with the XY parameter.

**Rotation Set** Runs the *BPARAMETER* command, Rotation parameter option with one grip specified, which adds a rotation parameter to the dynamic block definition. Automatically adds a rotation action associated with the rotation parameter.

**Flip Set** Runs the *BPARAMETER* command, Flip parameter option with one grip specified, which adds a flip parameter to the dynamic block definition. Automatically adds a flip action associated with the flip parameter.

**Visibility Set** Runs the *BPARAMETER* command, Visibility parameter option with one grip specified, which adds a visibility parameter to the dynamic block

definition and allows visibility states to be defined. No action is necessary with the visibility parameter.

**Lookup Set** Runs the *BPARAMETER* command, Lookup parameter option with one grip specified, which adds a lookup parameter to the dynamic block definition. Automatically adds a lookup action associated with the lookup parameter.

# -BEDIT

## Quick Reference

If you select a block in a drawing and enter **-bedit** at the command prompt, opens the selected block in the Block Editor (page 160). If nothing is selected, the following prompt is displayed:

Enter block name (page 168) or [? (page 168)]: *Enter a name or* **?**

### Block Name

Specifies the name of a block saved in the current drawing to open in the Block Editor or specifies the name of a new block to create.

### ?—List Previously Defined Blocks

Lists the block names in the text window.

Enter block(s) to list <*>: *Enter a name list or press* ENTER

# BGRIPSET

## Quick Reference

Creates, deletes, or resets grips associated with a parameter

Select a parameter in the block definition. Right-click in the Block Editor drawing area. Click Grip Display, and then click an option.
**bedit ➤ bgripset**

Specifies the number of grips displayed for a parameter. Also reset the position of grips for a parameter to their default location. You can only use the BGRIPSET command in the Block Editor (page 160).

Select parameter: *Select a parameter in the current dynamic block definition*
Enter number of grip objects for parameter or reset position [0/1/2/4/Reposition]: *Enter the number of grips to display for the parameter (the number of grips available will vary depending on the type of parameter you selected) or enter* **reposition** *to reposition the existing grips in the block definition to their default locations*

# BHATCH

## Quick Reference

Fills an enclosed area or selected objects with a hatch pattern or gradient fill

**bhatch**

The BHATCH command has been renamed to HATCH. If you enter **bhatch**, the Hatch and Gradient dialog box (page 618) is displayed. If you enter **-bhatch** or **-hatch**, the command prompts (page 631) are displayed.

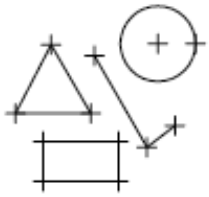
# BLIPMODE

## Quick Reference

Controls the display of marker blips

**blipmode** (or '**blipmode** for transparent use)
Enter mode [ON/OFF] <current>: *Enter* **on** *or* **off**, *or press* ENTER

When Blip mode is on, a temporary mark in the shape of a plus sign (+) appears where you specify a point. BLIPMODE is off by default.



objects drawn with BLIPMODE on

objects drawn with BLIPMODE off

To remove marker blips, use *REDRAW*, *REGEN*, *ZOOM*, *PAN*, or other commands that redraw or regenerate the drawing.

# BLOCK

## Quick Reference

Creates a block definition from objects you select

Draw 
Draw ➤ Block ➤ MakeAt the Command prompt, enter block.
**block**

The Block Definition dialog box (page 170) is displayed.

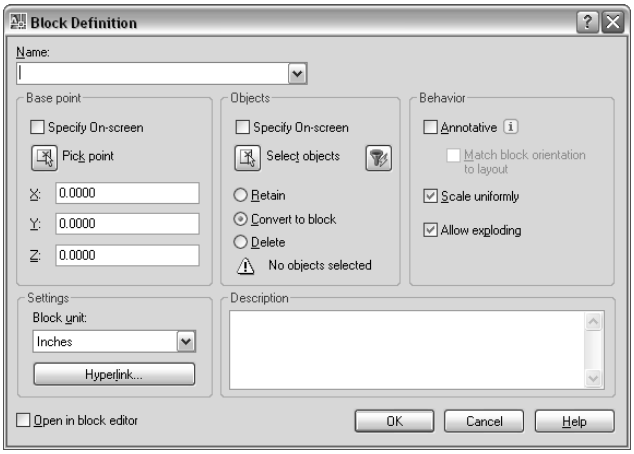
If you enter **-block** at the command prompt, options are displayed at the command prompt (page 173).

# Block Definition Dialog Box

## Quick Reference

Draw 
Draw ➤ Block ➤ MakeAt the Command prompt, enter block.
**block**

Defines and names a block.




### Name

Names the block. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used by the operating system or the program for other purposes.

The block name and definition are saved in the current drawing.

---

**NOTE** You cannot use DIRECT, LIGHT, AVE_RENDER, RM_SDB, SH_SPOT, and OVERHEAD as valid block names.

---

### Preview

If an existing block is selected under Name, displays a preview of the block.

### Base Point

Specifies an insertion base point for the block. The default value is 0,0,0.

**Specify On-Screen** Prompts you to specify the base point when the dialog box is closed.

**Pick Insertion Base Point** Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

**X** Specifies the *X* coordinate value.

**Y** Specifies the *Y* coordinate value.

**Z** Specifies the *Z* coordinate value.

### Objects

Specifies the objects to include in the new block and whether to retain or delete the selected objects or convert them to a block instance after you create the block.

**Specify On-Screen** Prompts you to specify the objects when the dialog box is closed.

**Select Objects** Closes the Block Definition dialog box temporarily while you select the objects for the block. When you finish selecting objects, press ENTER to redisplay the Block Definition dialog box.

**Quick Select** Displays the Quick Select dialog box (page 1152), which defines a selection set.

**Retain** Retains the selected objects as distinct objects in the drawing after you create the block.

**Convert to Block** Converts the selected objects to a block instance in the drawing after you create the block.

**Delete** Deletes the selected objects from the drawing after you create the block.

**Objects Selected** Displays the number of selected objects.

### Behavior

Specifies the behavior of the block.

**Annotative** Specifies that the block is annotative. Click the information icon to learn more about annotative objects.

**Match Block Orientation to Layout** Specifies that the orientation of the block references in paper space viewports matches the orientation of the layout. This option is unavailable if the Annotative option is cleared.

**Scale Uniformly** Specifies whether or not the block reference is prevented from being non-uniformly scaled.

**Allow Exploding** Specifies whether or not the block reference can be exploded.

### Settings

Specifies settings for the block.

**Block Unit** Specifies the insertion units for the block reference.

**Hyperlink** Opens the Insert Hyperlink dialog box (page 647), which you can use to associate a hyperlink with the block definition.

#### Description

Specifies the text description of the block.

#### Open in Block Editor

Opens the current block definition in the Block Editor (page 160) when you click OK.

# -BLOCK

## Quick Reference

If you enter **-block** at the command prompt, the following BLOCK command prompts are displayed.

Enter block name (page 173) or [? (page 175)]: *Enter a name or* **?**

#### Block Name

Names the block. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used by Microsoft Windows and the program for other purposes if the system variable *EXTNAMES* is set to 1.

If you enter the name of an existing block, you are prompted as follows:

Block *"NAME"* already exists. Redefine it? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

By redefining a block, you automatically update all references to that block. Attributes attached to existing block references remain unchanged in the drawing. However, new insertions of the block do not prompt for attributes unless the attribute definitions are included in the new block definition (use *ATTREDEF* to redefine blocks that contain attributes).

Specify insertion base point or [Annotative]: *Specify a point (1), enter* **a**, *or press* ENTER


The point specified as the base point is used for subsequent insertions of the block. Typically, a base point is the center of the block or its lower-left corner. The base point is also the point about which you can rotate the block during insertion. A block with 0 rotation is oriented according to the UCS in effect when it was created. Entering a 3D point inserts the block at a specific elevation. Omitting the *Z* coordinate uses the current elevation.

Enter **a** to create an block.

Create annotative block [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER

If you enter *yes*, the block becomes annotative.

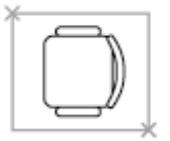
Match orientation to layout in paper space viewports [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER

If you enter *yes*, the block's orientation in paper space viewports will match the orientation of the layout.

If you specify the insertion base point, you are prompted to select the objects.

Select objects: *Use an object selection method*

The program defines a block using the objects selected, the insertion base point, and the name provided, and then erases the selected objects from the drawing. You can restore the deleted objects by entering the *OOPS* command immediately after BLOCK.



objects selected
as a block

The insertion base point becomes the origin of the block's coordinate system, which is parallel to the UCS in effect at the time that you define the block. When you insert the block into a drawing, its coordinate system is aligned parallel to the current UCS. Thus, you can insert a block at any orientation in space by setting the UCS first.

### ?—List Previously Defined Blocks

Lists the block names in the text window.

Enter block(s) to list <*>: *Enter a name list or press* ENTER

In the list, external references (xrefs) are indicated with the notation

Xref: resolved

In addition, externally dependent blocks (blocks in an xref) are indicated with the notation

xdep: *XREFNAME*

where *xrefname* is the name of an externally referenced drawing. The following terms are used in the list:

- *User Blocks:* Number of user-defined blocks in the list.
- *External References:* Number of xrefs in the list.
- *Dependent Blocks:* Number of externally dependent blocks in the list.
- *Unnamed Blocks:* Number of unnamed (anonymous) blocks in the drawing.

# BLOCKICON

## Quick Reference

Generates preview images for blocks displayed in DesignCenter

File ➤ Drawing Utilities ➤ Update Block IconsAt the Command prompt, enter blockicon.

**blockicon**

Enter block names <*>: *Specify block names, or press* ENTER *to update all blocks*

Use this command to generate icons for blocks created with an earlier release. You can enter a series of comma-delimited block names or wild-card characters. For example, enter **b1,?2**, to specify that block B1 and all two-character blocks ending with 2 should be updated.

After you enter the block names, a message is displayed describing the process as it proceeds. Press ESC at any time to stop.

# BLOOKUPTABLE

## Quick Reference

Displays or creates a lookup table for a dynamic block definition

**bedit ➤ baction**

Displays the Property Lookup Table dialog box (page 176).

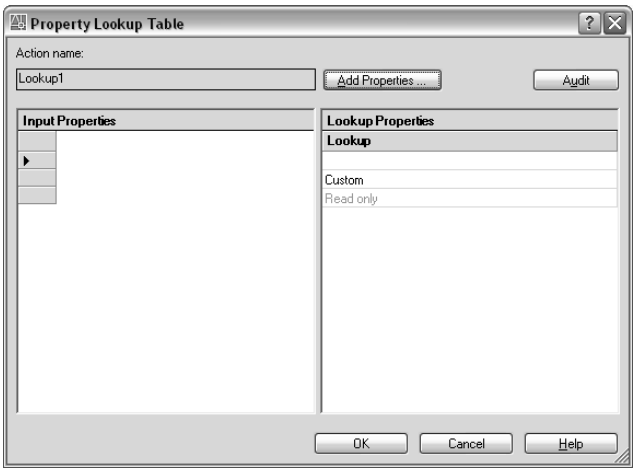
The current dynamic block definition must contain a lookup action and at least one lookup parameter in order for the Property Lookup Table dialog box to display. If a table is already defined for the lookup action, then that table is displayed in the dialog box. If no table is defined, one is created and displayed in the Property Lookup Table dialog box.

# Property Lookup Table Dialog Box

## Quick Reference

The Property Lookup Table dialog box provides a lookup table for the dynamic block definition. You use the lookup table to control the values available for a lookup parameter (lookup property). The lookup table also allows the values of lookup parameters to be controlled by the values of other parameters (input properties).

The lookup table assigns property values to the dynamic block reference based on how it is manipulated in a drawing. If Reverse Lookup is selected for a lookup property, the block reference displays a lookup grip. When the lookup grip is clicked in a drawing, a list of lookup properties is displayed. Selecting an option from that list will change the display of the dynamic block reference.




**Action Name** Displays the name of the lookup action associated with the table.

**Add Properties** Displays the Add Parameter Properties dialog box (page 178). You can add parameter properties to the lookup table.

**Audit** Checks the data in the table to ensure that each row is unique.

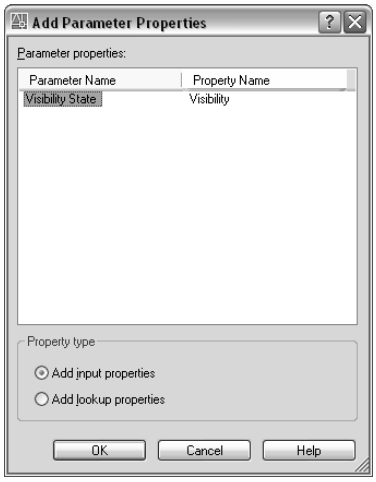
**Input Properties** Specifies the input properties for the dynamic block. Each parameter (except a lookup, alignment, or base point parameter) in the dynamic block can be displayed as a column under Input Properties. When a parameter's value matches a value in its corresponding input property column in the lookup table, the corresponding lookup property on the right side of the table is assigned to the block reference.

**Lookup Properties** Specifies the lookup properties for the dynamic block. Each lookup parameter in the dynamic block can be displayed as a column under Lookup Properties. When a parameter's value matches a value in its corresponding input property column in the lookup table, the corresponding lookup property on the right side of the table is assigned to the block reference. The drop-down control at the bottom of a lookup column is used to make the lookup property read-only or to allow reverse lookup. If all the rows in the table are unique, you can set the lookup property to allow reverse lookup. Allow Reverse Lookup enables the lookup property for a block reference to be set from a drop-down list that is displayed when the lookup grip is clicked in a drawing. Selecting an option from this list changes the block reference to match the corresponding input property values in the table.

# Add Parameter Properties Dialog Box

## Quick Reference

Adds parameter properties to a lookup table.




### Parameter Properties

Displays a list of parameters not yet added to the lookup table. Select a parameter and click OK to add it to the lookup table.

### Property Type

Specifies the type of property to add to the lookup table.

**Add Input Properties** When selected, the Parameter Properties list displays the available input property parameters that can be added to the lookup table.

**Add Lookup Properties** When selected, the Parameter Properties list displays the available lookup property parameters that can be added to the lookup table.

# BMPOUT

## Quick Reference

Saves selected objects to a file in device-independent bitmap format

**bmpout**

The Create Raster File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the file name in the dialog box.

Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A bitmap file that contains the objects you select is created. The file reflects what is displayed on the screen. Light glyphs that are displayed in the drawing appear in the new file, even if the Plot Glyph property of the lights is set to No.

**NOTE** When the *FILEDIA* system variable is set to 0 (Off), command prompts are displayed.

# BOUNDARY

## Quick Reference

Creates a region or a polyline from an enclosed area

Draw ➤ BoundaryAt the Command prompt, enter boundary.
**boundary**

The Boundary Creation dialog box (page 180) is displayed.

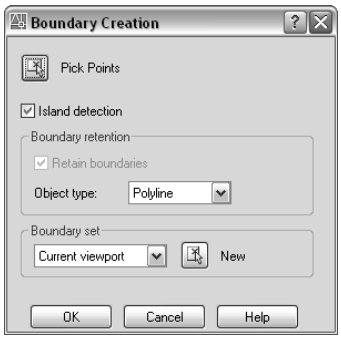
If you enter **-boundary** at the command prompt, options are displayed at the command prompt (page 181).

# Boundary Creation Dialog Box

## Quick Reference

Draw ➤ BoundaryAt the Command prompt, enter boundary.
**boundary**

Defines the object type, boundary set, and island detection method for creating a region or polyline using a specified point within an area enclosed by objects.




### Pick Points

Determines a boundary from existing objects that form an enclosed area around the specified point.

### Island Detection

Controls whether BOUNDARY detects internal closed boundaries, called islands.

### Object Type

Controls the type of the new boundary object. BOUNDARY creates the boundary as a region or a polyline object.

### Boundary Set

Defines the set of objects BOUNDARY analyzes when defining a boundary from a specified point.

**Current Viewport** Defines the boundary set from everything in the current viewport extents. Selecting this option discards any current boundary set.

**New** Prompts you to select the objects that define the boundary set. BOUNDARY includes only the objects that can be used to create a region or closed polyline when it constructs the new boundary set.

For more information about the options in this dialog box, see *HATCH*.

# -BOUNDARY

## Quick Reference

If you enter **-boundary** at the command prompt, the following BOUNDARY command prompts are displayed.

Specify internal point (page 181) or [Advanced options (page 181)]: *Specify a point or enter* **a**

### Internal Point

Creates a region or polyline from existing objects that form an enclosed area. Specify a point inside the area.

### Advanced Options

Sets the method BOUNDARY uses to create the boundary.

Enter an option [Boundary set/Island detection/Object type]: *Enter an option or press* ENTER *to return to the previous prompt*
**Boundary Set** Defines the set of objects BOUNDARY analyzes when it creates a boundary from a specified point. For information about defining a boundary set at the command prompt, see the *HATCH* command prompt option.

**Island Detection** Specifies whether BOUNDARY uses objects within the outermost boundary as boundary objects. For information about specifying island detection at the command prompt, see the *HATCH* command prompt option.

**Object Type** Specifies the type of object that BOUNDARY creates.

Enter type of boundary object [Region/Polyline] <*current*>: *Enter an option or press* ENTER

# BOX

## Quick Reference

Creates a 3D solid box

Modeling
Draw ➤ Modeling ➤ BoxAt the Command prompt, enter box.
**box**
3D Make panel, Box
Specify first corner or [Center (page 182)]: *Specify a point or enter***c** *for center*
Specify other corner or [Cube (page 183)/Length (page 184)]: *Specify the other corner of the box or enteran option*

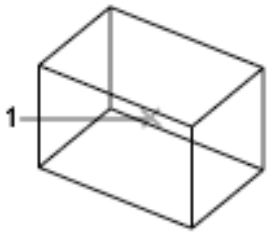
If the other corner of the box is specified with a *Z* value that differs from the first corner, then no height prompt is displayed.

Specify height or [2Point (page 184)] <default>: *Specify the height or enter* **2P** *for the 2 Point option*

Entering a positive value draws the height along the positive *Z* axis of the current UCS. Entering a negative value draws the height along the negative *Z* axis.

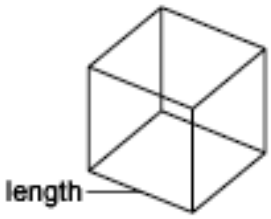
### Center

Creates the box by using a specified center point.




Specify center: *Specify a point (1)*
Specify other corner or [Cube/Length]: *Specify a point or enter an option*
Specify height or [2Point] <default>: *Specify the height or enter***2P** *for the 2 Point option*
**Cube** Creates a box with sides of equal length.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the box in the XY plane*

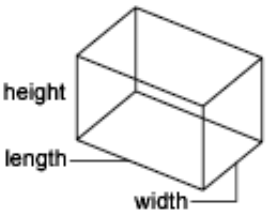



**Length** Creates a box with length, width, and height values you specify. The length corresponds to the *X* axis, the width to the *Y* axis, and the height to the *Z* axis.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the box in the XY plane*

Specify width: *Specify a distance*

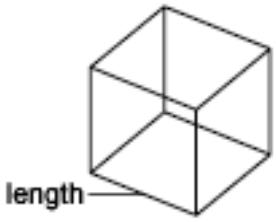
Specify height: *Specify a distance*




### Cube

Creates a box with sides of equal length.

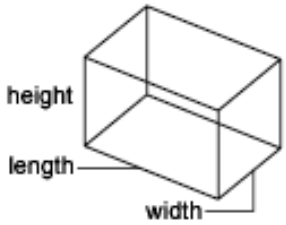
Specify length: *Enter a value, or pick a point to specify the length and the rotation of the box in the XY plane*



#### Length

Creates a box with length, width, and height values you specify. If you enter values, the length corresponds to the *X* axis, the width to the *Y* axis, and the height to the *Z* axis. If you pick a point to specify the length, you also specify the rotation in the *XY* plane.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the box in the XY plane*
Specify width: *Specify a distance*
Specify height: *Specify a distance*




#### 2Point

Specifies that the height of the box is the distance between the two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

# BPARAMETER

## Quick Reference

Adds a parameter with grips to a dynamic block definition

Block Editor 
**bedit ➤ bparameter**

You can use the BPARAMETER command only in the Block Editor (page 160). A parameter defines custom properties for the block reference. After you add a parameter, you must associate an action with the parameter to make the block dynamic.

Enter parameter type [Alignment (page 185)/Base (page 185)/pOint (page 186)/Linear (page 187)/Polar (page 189)/XY (page 192)/Rotation (page 194)/Flip (page 196)/Visibility (page 197)/looKup (page 198)] <last>: *Enter a parameter type*

### Alignment

Adds an alignment parameter to the current dynamic block definition. Because the alignment parameter affects the entire block, it is not necessary (or possible) to associate an action with the alignment parameter.

An alignment parameter defines an X and Y location and an angle. An alignment parameter always applies to the entire block and needs no action associated with it. An alignment parameter allows the block reference to automatically rotate around a point to align with other objects in the drawing. An alignment parameter affects the angle property of the block reference.

Specify base point of alignment or [Name]: *Specify a point or enter* **name**

When you specify the base point for the alignment parameter, an X is displayed in the Block Editor. When the command is completed, an alignment grip is added at this base point. The block reference automatically rotates about this point to align with another object in the drawing.

The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Alignment type = Perpendicular
Specify alignment direction or alignment type [Type] <Type>: *Specify an alignment direction or enter* **type**

The alignment direction specifies the direction of the grip and the angle of alignment for the block reference.

Enter alignment type [Perpendicular/Tangent] <Perpendicular>: *Specify an alignment type*
**Perpendicular** Specifies that the dynamic block reference aligns perpendicular to objects in a drawing.

**Tangent** Specifies that the dynamic block reference aligns tangent to objects in a drawing.

### Base

Adds a base point parameter to the current dynamic block definition. You do not associate any actions with a base point parameter. The base point parameter defines the base point for the dynamic block reference in relation to the geometry in the block. This provides a way to control the location of the base

point within the block reference when it is edited in a drawing. The base point parameter is generally included in a selection set of the block definition's actions.

Specify parameter location: *Specify a location*

Only one base point parameter is allowed in a dynamic block definition. If you try to add another base point parameter, the following alert is displayed.

Base point parameter already exists in block definition

### Point

Adds a point parameter to the current dynamic block definition and defines custom *X* and *Y* properties for the block reference. You associate a move or stretch action with a point parameter.

Specify parameter location or [Name/Label/Chain/Description/Palette]: *Specify a location or enter an option*
Specify label location: *Specify a location for the parameter label*
Enter number of grips [0/1] <1>: *Specify the number of grips for the parameter*

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter position property label <default>: *Enter a label for the position property or press* ENTER *to use the default label*

### Chain

Specifies the Chain Actions property for the parameter. The point parameter may be included in the selection set of an action that is associated with a different parameter. When that parameter is edited in a block reference, its

associated action may trigger a change in the values of other parameters included in the action's selection set.

If the Chain Actions property is set to Yes, this change triggers any actions associated with the point parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

If the Chain Actions property is set to No, the point parameter's associated actions are not triggered by the changes to the other parameter.

Evaluate associated actions when parameter is edited by another action? [Yes/No] <No>: *Enter* **y** *or press* ENTER

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Linear

Adds a linear parameter to the current dynamic block definition and defines a custom distance property for the block reference. You associate a move, scale, stretch, or array action with a linear parameter.

Specify start point or [Name/Label/Chain/Description/Base/Palette/Value set]: *Specify a start point for the parameter or enter an option*
Specify endpoint: *Specify an endpoint for the parameter*
Specify label location: *Specify a location for the parameter label*
Enter number of grips [0/1/2] <2>: *Specify the number of grips for the parameter*

If you specify one grip for the parameter, the grip is added to the endpoint of the parameter.

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter distance property label <default>: *Enter a label for the distance property or press* ENTER *to use the default label*

### Chain

Specifies the Chain Actions property for the parameter. The linear parameter may be included in the selection set of an action that is associated with a different parameter. When that parameter is edited in a block reference, its associated action may trigger a change in the values of other parameters included in the action's selection set.

If the Chain Actions property is set to Yes, this change triggers any actions associated with the linear parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

If the Chain Actions property is set to No, the linear parameter's associated actions are not triggered by the changes to the other parameter.

Evaluate associated actions when parameter is edited by another action? [Yes/No] <No>: *Enter* **y** *or press* ENTER

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Base

Specifies the Base Location property for the parameter.

Enter base location [Startpoint/Midpoint]: *Specify an option*
**Startpoint** Specifies that the start point of the parameter remains fixed when the endpoint of the parameter is edited in the block reference.

**Midpoint** Specifies a midpoint base location for the parameter. This midpoint is indicated by an X in the block definition. When you edit the linear parameter in the block reference, the midpoint of the parameter remains fixed, and the start point and endpoint of the parameter move simultaneously equal distances from the midpoint. For example, if you move the grip on the endpoint two units away from the midpoint, the start point simultaneously moves two units in the opposite direction.

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Value Set

Specifies a value set for the parameter. Limits the available values for the parameter in a block reference to the values specified in the set.

Enter distance value set type [None/List/Increment] <None>: *Specify a value set type* or press ENTER *to specify no value set type*
**List** Specifies a list of available values for the parameter in a block reference.

Enter list of distance values (separated by commas): *Specify a list of values separated by commas*

**Increment** Specifies a value increment and minimum and maximum values for the parameter in the block reference.

Enter distance increment: *Specify an increment value for the parameter*

Enter minimum distance: *Specify a minimum distance value for the parameter*

Enter maximum distance: *Specify a maximum distance value for the parameter*

### Polar

Adds a polar parameter to the current dynamic block definition. Defines custom distance and angle properties for the block reference. You associate a move, scale, stretch, polar stretch, or array action with a polar parameter.

Specify base point or [Name/Label/Chain/Description/Palette/Value set]: *Specify a start point for the parameter or enter an option*
Specify endpoint: *Specify an endpoint for the parameter*
Specify label location: *Specify a location for the parameter label*
Enter number of grips [0/1/2] <2>: *Specify the number of grips for the parameter*

If you specify one grip for the parameter, the grip is added to the endpoint of the parameter.

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter distance property label <default>: *Enter a label for the distance property or press* ENTER *to use the default label*
Enter angle property label <default>: *Enter a label for the angle property or press* ENTER *to use the default label*

### Chain

Specifies the Chain Actions property for the parameter. The polar parameter may be included in the selection set of an action that is associated with a different parameter. When that parameter is edited in a block reference, its associated action may trigger a change in the values of other parameters included in the action's selection set.

If the Chain Actions property is set to Yes, this change triggers any actions associated with the polar parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

If the Chain Actions property is set to No, the polar parameter's associated actions are not triggered by the changes to the other parameter.

Evaluate associated actions when parameter is edited by another action? [Yes/No] <No>: *Enter* **y** *or press* ENTER

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Value Set

Specifies a value set for the parameter. Limits the available values for the parameter in a block reference to the values specified in the set.

Enter distance value set type [None/List/Increment] <None>: *Specify a distance value set type* or press ENTER *to specify no value set type*
Enter angle value set type [None/List/Increment] <None>: *Specify an angle value set type* or press ENTER *to specify no value set type*
**List** Specifies a list of available values for the parameter in a block reference.

Enter list of distance values (separated by commas): *Specify a list of values separated by commas*

Enter list of angle values (separated by commas): *Specify a list of values separated by commas*

**Increment** Specifies a value increment and minimum and maximum values for the parameter distance in the block reference.

Enter distance increment: *Specify an increment value for the parameter distance*

Enter minimum distance: *Specify a minimum value for the parameter distance*

Enter maximum distance: *Specify a maximum value for the parameter distance*

Specifies a value increment and minimum and maximum values for the parameter angle in the block reference.

Enter angle increment: *Specify an increment value for the parameter angle*

Enter minimum angle: *Specify a minimum value for the parameter angle*

Enter maximum angle: *Specify a maximum value for the parameter angle*

### XY

Adds an XY parameter to the current dynamic block definition and defines custom horizontal and vertical distance properties for the block reference. You associate a move, scale, stretch, or array action with an XY parameter.

Specify base point or [Name/Label/Chain/Description/Palette/Value set]: *Specify a base point for the parameter or enter an option*
Specify endpoint: *Specify an endpoint for the parameter*
Enter number of grips [0/1/2/4] <1>: *Specify the number of grips for the parameter*

If you specify one grip for the parameter, the grip is added to the endpoint of the parameter. If you specify two grips for the parameter, a grip is added to the base point and endpoint of the parameter. If you specify four grips for the parameter, a grip is added to all four corners of the parameter.

#### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

#### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter horizontal distance property label <default>: *Enter a label for the horizontal distance property or press* ENTER *to use the default label*
Enter vertical distance property label <default>: *Enter a label for the vertical distance property or press* ENTER *to use the default label*

#### Chain

Specifies the Chain Actions property for the parameter. The XY parameter may be included in the selection set of an action that is associated with a different parameter. When that parameter is edited in a block reference, its associated action may trigger a change in the values of other parameters included in the action's selection set.

If the Chain Actions property is set to Yes, this change triggers any actions associated with the XY parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

If the Chain Actions property is set to No, the XY parameter's associated actions are not triggered by the changes to the other parameter.

Evaluate associated actions when parameter is edited by another action? [Yes/No] <No>: *Enter* **y** *or press* ENTER

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter horizontal distance property description: *Enter a description for the horizontal distance property of the parameter*
Enter vertical distance property description: *Enter a description for the vertical distance property of the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Value Set

Specifies a value set for the parameter. Limits the available values for the parameter in a block reference to the values specified in the set.

Enter horizontal value set type [None/List/Increment] <None>: *Specify a value set type* or press ENTER *to specify no value set type*
Enter vertical value set type [None/List/Increment] <None>: *Specify a value set type* or press ENTER *to specify no value set type*
**List** Specifies a list of available values for the parameter in a block reference.

Enter list of distance values (separated by commas): *Specify a list of values separated by commas*

**Increment** Specifies a value increment and minimum and maximum values for the parameter in the block reference.

Enter distance increment: *Specify an increment value for the parameter*

Enter minimum distance: *Specify a minimum distance value for the parameter*

Enter maximum distance: *Specify a maximum distance value for the parameter*

### Rotation

Adds a rotation parameter to the current dynamic block definition and defines a custom angle property for the block reference. Only a rotate action can be associated with a rotation parameter.

Specify base point or [Name/Label/Chain/Description/Palette/Value set]: *Specify a base point for the parameter or enter an option*
Specify radius of parameter: *Specify a radius for the parameter*
Specify default rotation angle or [Base angle] <0>: *Specify a base angle for the parameter*
Specify label location: *Specify a location for the parameter label*
Enter number of grips [0/1] <1>: *Specify the number of grips for the parameter*

If you specify one grip for the parameter, the grip is added to the key point on the radius of the parameter.

**Base Angle** Specifies a base angle for the parameter and places the grip for the parameter at this angle.

Specify base angle <0>: *Specify a base angle for the parameter or press* ENTER

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter rotation property label <default>: *Enter a label for the rotation property or press* ENTER *to use the default label*

### Chain

Specifies the Chain Actions property for the parameter. The rotation parameter may be included in the selection set of an action that is associated with a different parameter. When that parameter is edited in a block reference, its associated action may trigger a change in the values of other parameters included in the action's selection set.

If the Chain Actions property is set to Yes, this change triggers any actions associated with the rotation parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

If the Chain Actions property is set to No, the rotation parameter's associated actions are not triggered by the changes to the other parameter.

Evaluate associated actions when parameter is edited by another action? [Yes/No] <No>: *Enter* **y** *or press* ENTER

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Value Set

Specifies a value set for the parameter. Limits the available values for the parameter in a block reference to the values specified in the set.

Enter angle value set type [None/List/Increment] <None>: *Specify an angle value set type* or press ENTER *to specify no value set type*
**List** Specifies a list of available values for the parameter in a block reference.

Enter list of angle values (separated by commas): *Specify a list of values separated by commas*

**Increment** Specifies a value increment and minimum and maximum values for the parameter angle in the block reference.

Enter angle increment: *Specify an increment value for the parameter angle*

Enter minimum angle: *Specify a minimum value for the parameter angle*

Enter maximum angle: *Specify a maximum value for the parameter angle*

### Flip

Adds a flip parameter to the current dynamic block definition. Defines a custom flip property for the block reference. A flip parameter flips objects. In the Block Editor, a flip parameter displays as a reflection line. Objects can be flipped about this reflection line. A flip parameter displays a value that shows if the block reference has been flipped or not. You associate a flip action with a flip parameter.

Specify base point of reflection line or [Name/Label/Description/Palette]: *Specify a base point for the reflection line or enter an option*
Specify endpoint of reflection line: *Specify an endpoint for the reflection line*
Specify label location: *Specify a location for the parameter label*
Enter number of grips [0/1] <1>: *Specify the number of grips for the parameter*

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter Flip property label <default>: *Enter a label for the flip property or press* ENTER *to use the default label*
Specify values for items in the flip property dropdown control:
Enter flip property value for an unflipped state <Not Flipped>: *Enter a property value to display when the parameter is unflipped or press* ENTER *to use the default label*
Enter flipped property value for a flipped state <Flipped>: *Enter a property value to display when the parameter is flipped or press* ENTER *to use the default label*

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Visibility

Adds a visibility parameter to the current dynamic block definition and defines a custom visibility property for the block reference. A visibility parameter allows you to create visibility states and to control the visibility of objects in the block. A visibility parameter always applies to the entire block and needs no action associated with it.

Specify parameter location or [Name/Label/Description/Palette]: *Specify a location for the parameter or enter an option*
Enter number of grips [0/1] <1>: *Specify the number of grips for the parameter*

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter visibility property label <default>: *Enter a label for the visibility property or press* ENTER *to use the default label*

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

### Lookup

Adds a lookup parameter to the current dynamic block definition and defines custom lookup properties for the block reference. A lookup parameter defines a custom property that you can specify or set to evaluate to a value from a list or table you define. You associate a lookup action with a lookup parameter. Each lookup parameter you add to the block definition can be added as a column in the Property Lookup Table dialog box (page 176).

Specify parameter location or [Name/Label/Description/Palette]: *Specify a location for the parameter or enter an option*
Enter number of grips [0/1] <1>: *Specify the number of grips for the parameter*

### Name

Specifies the parameter name. The parameter name is displayed in the Properties palette when you select the parameter in the Block Editor.

Enter parameter name <default>: *Enter a name for the parameter or press* ENTER *to use the default name*

### Label

Specifies the parameter label, which defines the custom property name added to the block. The parameter label is displayed in the Properties palette as a Custom property when you select the block reference in a drawing. In the Block Editor, the parameter label is displayed next to the parameter.

Enter lookup property label <default>: *Enter a label for the lookup property or press* ENTER *to use the default label*

### Description

Specifies the description for the custom property name (parameter label). This description is displayed in the Properties palette when you select the parameter in the Block Editor. In a drawing, when you select the custom property name (parameter label) for the block reference in the Properties palette, the description is displayed at the bottom of the Properties palette.

Enter property description: *Enter a description for the parameter*

### Palette

Specifies whether or not the parameter label is displayed in the Properties palette when the block reference is selected in a drawing.

Display property in Properties palette? [Yes/No] <Yes>: *Enter* **n** *or press* ENTER

# BREAK

## Quick Reference

Breaks the selected object between two points

Modify 

Modify ➤ BreakAt the Command prompt, enter break.

**break**

Select object: *Use an object selection method, or specify the first break point (1) on an object*

The prompts that are displayed next depend on how you select the object. If you select the object by using your pointing device, the program both selects the object and treats the selection point as the first break point. At the next prompt you can continue by specifying the second point or overriding the first point.

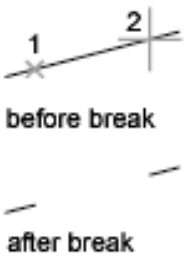
Specify second break point or [First point]: *Specify the second break point (2) or enter* **f**

**Second Break Point** Specifies the second point to use to break the object.

**First Point** Overrides the original first point with the new point that you specify.

Specify first break point:

Specify second break point:

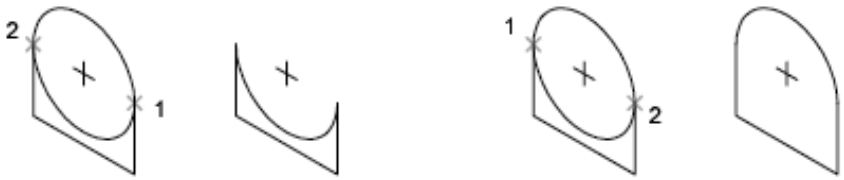



The portion of the object is erased between the two points that you specify. If the second point is not on the object, the nearest point on the object is selected; therefore, to break off one end of a line, arc, or polyline, specify the second point beyond the end to be removed.

To split an object in two without erasing a portion, enter the same point for both the first and second points. You can do this by entering **@** to specify the second point.

Lines, arcs, circles, polylines, ellipses, splines, donuts, and several other object types can be split into two objects or have one end removed.

The program converts a circle to an arc by removing a piece of the circle starting counterclockwise from the first to the second point.




# BREP

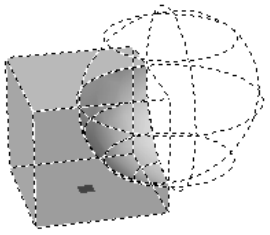
## Quick Reference

Removes the history from 3D solid primitives and composite solids

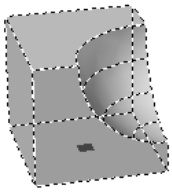
**brep**
Select objects: *Use an object selection method and press* ENTER *when you finish*

The history of the selected solids is removed. The solid no longer retains a history of the original parts from which the solid was created, and thus, its original parts cannot be selected and modified.

If you use BREP on a solid primitive, the solid primitive is no longer a true primitive and cannot be manipulated as such (through grips and the Properties palette).



Composite solid with Show History set to Yes



Composite solid with history removed and Show History set to Yes

# BROWSER

## Quick Reference

Launches the default web browser defined in your system's registry

**Web toolbar:** 

**browser**
Enter Web location (URL) <*current*>: *Press* ENTER *or enter a new location; you don't need to enter http:// before the location*

Pressing ENTER displays your web browser, which automatically connects to the location you specify. Because BROWSER does not append "http://" to web locations, you can specify an FTP or file location to your default web browser.

# BSAVE

## Quick Reference

Saves the current block definition

Block Editor 
**bedit ➤ bsave**

Saves changes to the current block definition.

You can only use the BSAVE command in the Block Editor (page 160).

# BSAVEAS

## Quick Reference

Saves a copy of the current block definition under a new name

Block Editor 
**bedit ➤ bsaveas**

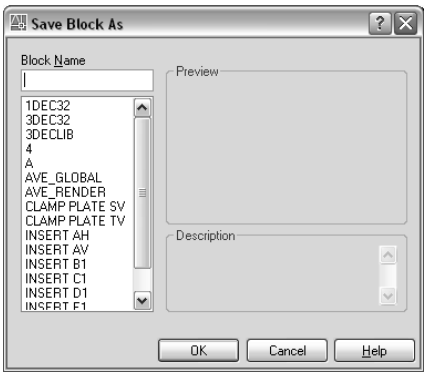
Displays the Save Block As dialog box (page 202).

You can only use the BSAVEAS command in the Block Editor (page 160).

# Save Block As Dialog Box

## Quick Reference

Saves a copy of the current block definition under a new name.




**Block Name** Specifies a new name under which to save a copy of the current block definition.

**Block List** Displays a list of block definitions that are saved in the current drawing.

**Preview** Displays a preview of the selected block definition.

**Description** Displays the block definition description specified in the Block area of the Properties palette in the Block Editor.

# BVHIDE

## Quick Reference

Makes objects invisible in the current visibility state or all visibility states in a dynamic block definition

Block Editor 
Select objects to hide for visibility states. Right-click in the Block Editor drawing area. Click Object Visibility ➤ Hide for Current State or click Object Visibility ➤ Hide for All States
**bedit ➤ bvhide**

Allows you to make objects invisible for the current visibility state. You can only use the BVHIDE command in the Block Editor (page 160).

Select objects to hide: *Select objects to hide for the current visibility state or all visibility states*

Hide for current state or all visibility states [Current/All] <Current>: *Enter* **current***to hide the selected objects for the current visibility state or enter***all***to hide the selected objects for all visibility states in the block definition*

# BVSHOW

## Quick Reference

Makes objects visible in the current visibility state or all visibility states in a dynamic block definition

Block Editor 
Select objects to make visible for visibility states. Right-click in the Block Editor drawing area. Click Object Visibility ➤ Show for Current State or click Object Visibility ➤ Show for All States
**bedit ➤ bvshow**

Allows you to make objects visible for visibility states. You can only use the BVSHOW command in the Block Editor (page 160).

Select objects to make visible: *Select objects to make visible for the current visibility state or all visibility states*
Make visible for current state or all visibility states [Current/All] <Current>: *Enter* **current** *to make the selected objects visible for the current visibility state or enter* **all** *to make the selected objects visible for all visibility states in the block definition*

# BVSTATE

## Quick Reference

Creates, sets, or deletes a visibility state in a dynamic block

**bedit ➤ bvstate**

Displays the Visibility States dialog box (page 205). You can only use the BVSTATE command in the Block Editor (page 160) after a visibility parameter has been added to the block definition.

# Visibility States Dialog Box

## Quick Reference

Creates or modifies visibility states. Through buttons in the dialog box or right-click menu options, you can

- Create new visibility states
- Set any visibility state as the current state
- Rename visibility states
- Delete visibility states
- Move visibility states up or down in the list


**Visibility States** Lists the available visibility states for the current dynamic block definition. The order of this list is reflected in the dynamic block reference when the grip is clicked to display the list of visibility states. The state at the top of the list is the default state for the block reference.

**Set Current** Sets the selected visibility state as the current state to display in the Block Editor. Does not change the default visibility state that is displayed when the block is inserted in a drawing, nor does it change the displayed visibility state for block references already inserted and edited in the drawing.

**New** Displays the New Visibility State dialog box. (page 206)

**Rename** Renames the selected visibility state.

**Delete** Deletes the selected visibility state.

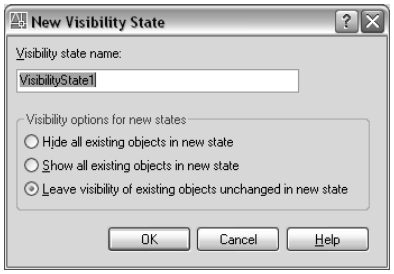
**Move Up** Moves the selected visibility state up in the list.

**Move Down** Moves the selected visibility state down in the list.

# New Visibility State Dialog Box

## Quick Reference

Creates a new visibility state.




**Visibility State Name** Specifies the name for the new visibility state.

**Visibility Options for New States** Displays options for the new visibility state.

**Hide All Existing Objects in New State** Specifies that all objects in the block definition will be hidden in the new visibility state.

**Show All Existing Objects in New State** Specifies that all objects in the block definition will be shown in the new visibility state.

**Leave Visibility of Existing Objects Unchanged in New State** Specifies that the visibility of objects in the new visibility state will be the same as in the current visibility state on which the new state is based.

**OK** Creates a new visibility state.

# -BVSTATE

## Quick Reference

If you enter **-bvstate** at the command prompt, the following BVSTATE command prompts are displayed.

Current visibility state: *<Name of current visibility state>*

Enter an option [New (page 207)/Set (page 207)/Delete (page 207)] <New>: Enter an option or press ENTER

### New

Creates a new visibility state.

Enter name of new visibility state: *Enter a name for the new visibility state*
Enter object visibility for new state [Hide all/Show all/Current visibility] <Current visibility>: *Enter object visibility for new state*
**Hide All** Specifies that all objects in the block definition will be hidden in the new visibility state.

**Show All** Specifies that all objects in the block definition will be shown in the new visibility state.

**Current Visibility** Specifies that the visibility of objects in the new visibility state will be the same as in the current visibility state on which the new state is based.

### Set

Specifies the visibility state to set as current.

Enter visibility state name to make current or [?]: *Enter the name of the visibility state to set as the current state*

### Delete

Deletes a visibility state.

Enter visibility state name(s) to delete: *Enter the name(s) of the visibility states to delete*

# C Commands

# 4

**In this chapter**

- CONVERTPSTYLES
- CONVTOSOLID
- CONVTOSURFACE
- COPY
- COPYBASE
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- COPYHIST
- COPYLINK
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- CUIEXPORT
- CUIIMPORT
- CUILOAD
- CUIUNLOAD
- CUSTOMIZE
- CUTCLIP
- CYLINDER

# CAL

## Quick Reference

Evaluates mathematical and geometric expressions

**cal** (or '**cal** for transparent use)

CAL is an online geometry calculator that evaluates point (vector), real, or integer expressions. The expressions can access existing geometry using the object snap functions such as CEN, END, and INS.

You can insert AutoLISP® variables into the arithmetic expression and assign the value of the expression back to an AutoLISP variable.

You can use these arithmetic and vector expressions in any command that expects points, vectors, or numbers.

# Understand Syntax of Expressions

## Quick Reference

CAL evaluates expressions according to standard mathematical rules of precedence:

- Expressions in parentheses first, starting with the innermost set
- Operators in standard order: exponents first, multiplication and division second, and addition and subtraction last
- Operators of equal precedence from left to right

### Numeric Expressions

Numeric expressions are real integer numbers and functions combined with the operators in the following table.

**Numeric operators**

| Operator | Operation |
|---|---|
| ( ) | Groups expressions |

**Numeric operators**

| Operator | Operation |
|---|---|
| ^ | Indicates exponentiation |
| * , / | Multiplies, divides |
| +, - | Adds, subtracts |

The following are examples of numeric expressions:

**3**

**3 + 0.6**

**(5.8^2) + PI**

### Vector Expressions

A vector expression is a collection of points, vectors, numbers, and functions combined with the operators in the following table.

**Vector operators**

| Operator | Operation |
|---|---|
| ( ) | Groups expressions |
| & | Determines the vector product of vectors (as a vector)<br>[a,b,c]&[x,y,z] = [ (b*z) - (c*y) , (c*x) - (a*z) , (a*y) - (b*x) ] |
| * | Determines the scalar product of vectors (as a real number)<br>[a,b,c]*[x,y,z] = ax + by + cz |
| *, / | Multiplies, divides a vector by a real number<br>a*[x,y,z] = [a*x,a*y,a*z] |
| + , - | Adds, subtracts vectors (points)<br>[a,b,c] + [x,y,z] = [a+x,b+y,c+z] |

The following are examples of vector expressions:

**A+[1,2,3]** provides the point positioned [1,2,3] units relative to point A.

The expression

**[2<45<45] + [2<45<0] - [1.02, 3.5, 2]**

adds two points and subtracts a third point. The first two points are in spherical coordinates.

# Format Feet and Inches

## Quick Reference

Enter feet and inches using the following format:

**feet'-inches"** or **feet' inches"** or **feet'inches"**

You can separate feet, inches, and fractional inches with a dash, a space, or nothing. You can use any of the following syntax cases to enter valid feet-inch formatted values:

- 5' or 60"
- 5'-9" or 5' 9" or 5'9"
- 5'-1/2" or 5' 1/2" or 5'1/2"
- 5'-9-1/2" or 5' 9-1/2" or 5'9-1/2"
- 5'-9 1/2" or 5' 9 1/2" or 5'9 1/2"

To designate inches for linear calculations, entering double quotes (") is optional. For example, instead of entering 5'9-1/2", you could enter 5'9-1/2.

---

**WARNING** With imperial units, CAL interprets a minus or a dash (-) as a unit separator rather than a subtraction operation. To specify subtraction, include at least one space before or after the minus sign. For example, to subtract 9" from 5', enter **5' -9"** rather than **5'-9"**.

---

# Format Angles

## Quick Reference

The default units for angles are decimal degrees. Enter angles using the following format:

*<degrees>***d***<minutes>'<seconds>"*

You must enter **0d** when entering an angle that is less than 1 degree (minutes and seconds only). You can omit the minutes or seconds if they are zero.

Enter a number followed by **r** to enter angles in radians. Enter a number followed by **g** to enter angles in grads.

The following examples show ways of entering angles:

**5d10'20"**

**0d10'20"**

**124.6r**

**14g**

Angles entered in any format are converted to decimal degrees.

Pi radians is equal to 180 degrees, and 100 grads is equal to 90 degrees.

# Use Points and Vectors

## Quick Reference

Both points and vectors are pairs or triples of real numbers. A point defines a location in space, but a vector defines a direction (or translation) in space.

Some CAL functions, such as **pld** and **plt**, return a point. Other functions, such as **nor** and **vec**, return a vector.

### Formatting Points and Vectors

A point or vector is a set of three real expressions enclosed in brackets ([ ]):
*[r1,r2,r3]*

The notation *p1*, *p2*, and so forth designates points. The notation *v1*, *v2*, and so forth designates vectors. In drawings, points are displayed as dots, and vectors are displayed as lines with arrows.

CAL supports points expressed in all formats.

**Point formats**

| Coordinate system | Point format |
|---|---|
| Polar | [dist<angle] |
| Cylindrical | [dist<angle,z] |
| Spherical | [dist<angle1<angle2] |
| Relative | Uses the @ prefix [@x,y,z] |
| WCS (instead of UCS) | Uses the * prefix [*x,y,z] |

You can omit the following components of a point or vector: coordinate values of zero and a comma immediately preceding the right bracket (]).

The following are valid points:

**[1,2]** is the same as **[1,2,0]**

**[,,3]** is the same as **[0,0,3]**

**[ ]** is the same as **[0,0,0]**

In the following example, the point is entered in the relative spherical coordinate system with respect to the (WCS). The distance is 1+2=3; the angles are 10+20=30 degrees and 45 degrees, 20 minutes.

**[@*1+2<10+20<45d20"]**

The following is a valid point that contains arithmetic expressions as its components:

**[2*(1.0+3.3),0.4-1.1,2*1.4]**

The following example uses the Endpoint object snap and the vector [2,0,3] to calculate a point that is offset from a selected endpoint:

**end + [2,,3]**

The calculated point is offset 2 units in the *X* direction and 3 units in the *Z* direction relative to the selected endpoint.

# Use AutoLISP Variables

## Quick Reference

You can use AutoLISP variables within arithmetic expressions. The variables must be one of the following types: real, integer, or 2D or 3D point (vector).

This example defines a point positioned 5 units in the *X* direction and 1 unit in the *Y* direction from the point stored in AutoLISP variable `A`.

**A+[5,1]**

If you enter an AutoLISP variable with a name containing a character with special meaning in CAL, such as +, -, *, or /, enclose the variable name in apostrophes ('), for example:

'**number-of-holes**'

### Assigning Values to AutoLISP Variables

To assign a value to an AutoLISP variable, precede the arithmetic expression with the variable name and the equal sign (=). Later, you can use the value of this variable for other calculations.

This example saves the values of two expressions in AutoLISP variables `P1` and `R1`.

Command: **cal**
>> Expression: **P1=cen+[1,0]**
>> Select entity for CEN snap: *Select a circle or an arc*
Command: **cal**
>> Expression: **R1=dist(end,end)/3**
>> Select entity for END snap: *Select an object with an endpoint*

This example uses the values of variables `P1` and `R1`:

Command: **circle**
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: '**cal**
>> Expression: **P1+[0,1]**
Specify radius of circle or [Diameter] <*last*>: '**cal**
>> Expression: **R1+0.5**

## Use System Variables in Calculations

### Quick Reference

You can use the `getvar` function to read the value of a system variable.

The syntax is

```
getvar(variable_name)
```

The following example uses `getvar` to obtain the point that is the center of the view in the current viewport.

**getvar(viewctr)**

With this method, you can also access the user system variables, USERI1-5 and USERR1-5. For example, to retrieve the value stored in USERR2, enter the following:

**getvar(userr2)**

## Convert Units of Measurement

### Quick Reference

The `cvunit` function converts either a number or a point from one unit of measurement to another. See the *acad.unt* file for a list of units that you can convert. The syntax is

```
cvunit(value, from_unit, to_unit)
```

The following example converts the value 1 from inches to centimeters:

**cvunit(1,inch,cm)**

# Use Standard Numeric Functions

## Quick Reference

CAL supports the standard numeric functions in the following table.

**Numeric functions**

| Function | Description |
|---|---|
| sin(*angle*) | Sine of the angle |
| cos(*angle*) | Cosine of the angle |
| tang(*angle*) | Tangent of the angle |
| asin(*real*) | Arcsine of the number; the number must be between -1 and 1 |
| acos(*real*) | Arccosine of the number; the number must be between -1 and 1 |
| atan(*real*) | Arctangent of the number |
| ln(*real*) | Natural log of the number |
| log(*real*) | Base-10 log of the number |
| exp(*real*) | Natural exponent of the number |
| exp10(*real*) | Base-10 exponent of the number |
| sqr(*real*) | Square of the number |
| sqrt(*real*) | Square root of the number; the number must be nonnegative |
| abs(*real*) | Absolute value of the number |
| round(*real*) | Number rounded to the nearest integer |

| Numeric functions | |
|---|---|
| **Function** | **Description** |
| trunc(*real*) | Integer portion of the number |
| r2d(*angle*) | Angles in radians converted to degrees; for example, **r2d(pi)** converts the pi radians to 180 degrees |
| d2r(*angle*) | Angles in degrees converted to radians; for example, **d2r(180)** converts 180 degrees to radians and returns the value of pi |
| pi | The constant pi |

# Calculate a Vector from Two Points

## Quick Reference

The functions **vec** and **vec1** calculate a vector from two points.

**vec(p1,p2)** Provides the vector from point *p1* to point *p2*.

**vec1(p1,p2)** Provides the unit vector from point *p1* to point *p2*.

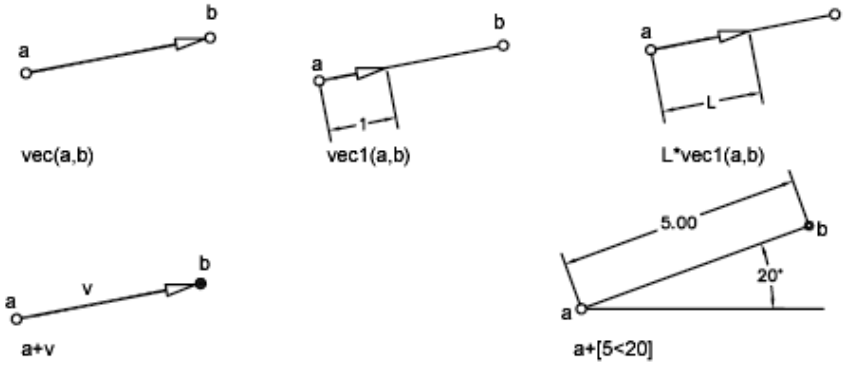
The following example uses CAL to move selected objects 3 units in the direction from the center of one selected circle to the center of another selected circle:

Command: **move**
Select objects
Specify base point or displacement: '**cal**
>> Expression: **3*vec1(cen,cen)**
Select entity for CEN snap: *Specify a circle or an arc*
Specify second point of displacement or <use first point as displacement>:
*Specify a point or press* ENTER

The following examples illustrate the meaning of vector and point calculations.

**Examples of vector and point calculations**

| Expression | Meaning |
|---|---|
| vec(*a,b*) | Determines vector translation from point *a* to point *b*. |
| vec1(*a,b*) | Determines unit vector direction from point *a* to point *b*. |
| L*vec1(*a,b*) | Determines vector of length *L* in the direction from point *a* to point *b*. |
| *a*+*v* | Determines point *b*, which is a translation of the point *a* through vector *v*. |
| *a*+[5<20] | Determines point *b* positioned 5 units away from point *a* under the angle of 20 degrees. Note that [5<20] is a vector in polar coordinates. |




# Calculate the Length of a Vector

## Quick Reference

The **abs** function calculates the length of a vector.

**abs(v)** Calculates the length of vector *v*, a nonnegative real number.

In spherical coordinates (*dist<ang<ang*), the *dist* is the length of the vector.

The following example calculates the length of the vector [1,2,3]:

**abs([1,2,3])**

## Obtain a Point by Cursor

### Quick Reference

To enter a point using the pointing device, use the **cur** function. The program prompts you to specify a point and uses the coordinate values of the point in the expression. The point coordinate values are expressed in terms of the current UCS. The **cur** function sets the value of the *LASTPOINT* system variable.

The following example adds the vector [3.6,2.4,0]—the result of 1.2*[3,2]—to the point you select. This expression produces a point that is offset from the selected point.

**cur+1.2*[3,2]**

## Obtain the Last-Specified Point

### Quick Reference

Use the @ character in the expression to obtain the coordinate of the last point, as shown in the following example:

Command: **line**
Specify first point: '**cal**
>> Expression: **cen+[0,1]**
>> Select entity for CEN snap: *Select a circle or an arc*
Specify next point or [Close/Undo]: '**cal**
>> Expression: **@+3*vec1(cen,cen)**

The first point of the line is 1 unit in the *Y* direction from the center of the first selected circle. The second point of the line is 3 units away from the first point. The direction of the line is from the center of the first selected circle to the center of the second selected circle.

# Use Snap Modes in Arithmetic Expressions

## Quick Reference

You can use Snap modes as parts of arithmetic expressions. The program prompts you to select an object and returns the coordinate of the appropriate snap point. Using arithmetic expressions with Snap modes greatly simplifies entering coordinates relative to other objects.

When you use these Snap modes, enter only the three-character name. For example, when you use the Center Snap mode, enter **cen**. CAL Snap modes set the value of the *LASTPOINT* system variable.

**CAL Snap modes**

| Abbreviation | Snap mode |
|---|---|
| END | ENDPOINT |
| INS | INSERT |
| INT | INTERSECTION |
| MID | MIDPOINT |
| CEN | CENTER |
| NEA | NEAREST |
| NOD | NODE |
| QUA | QUADRANT |
| PER | PERPENDICULAR |
| TAN | TANGENT |

The following example uses the Center and Endpoint Snap modes in a CAL expression:

**(cen+end)/2**

CAL prompts for a circle or arc and an object. It then determines the midpoint between the center of the circle or arc and the end of the selected object.

Using the Midpoint Snap mode, in the following example CAL prompts for an object and returns a point 1 unit in the *Y* direction from the midpoint of the selected object:

**mid+[,1]**

The following example uses the Endpoint Snap mode to calculate the centroid of a triangle defined by three endpoints:

**(end+end+end)/3**

# Convert Points Between UCS and WCS

## Quick Reference

Normally, the program assumes all coordinates to be relative to the current UCS. The following functions convert points between UCS and WCS.

**w2u(p1)** Converts point *p1* expressed in the WCS to the current UCS.

**u2w(p1)** Converts point *p1* expressed in the current UCS to the WCS.

You can use **w2u** to find the WCS origin in terms of the current UCS:

**w2u([0,0,0])**

### Filtering the X, Y, and Z Components of a Point or Vector

The following functions filter the *X*, *Y*, and *Z* components of a point or vector.

**Point-filter functions**

| Function | Description |
|---|---|
| xyof(*p1*) | *X* and *Y* components of a point; *Z* component is set to 0.0 |
| xzof(*p1*) | *X* and *Z* components of a point; *Y* component is set to 0.0 |
| yzof(*p1*) | *Y* and *Z* components of a point; *X* component is set to 0.0 |

**Point-filter functions**

| Function | Description |
|---|---|
| xof(*p1*) | *X* component of a point; *Y* and *Z* components are set to 0.0 |
| yof(*p1*) | *Y* component of a point; *X* and *Z* components are set to 0.0 |
| zof(*p1*) | *Z* component of a point; *X* and *Y* components are set to 0.0 |
| rxof(*p1*) | *X* component of a point |
| ryof(*p1*) | *Y* component of a point |
| rzof(*p1*) | *Z* component of a point |

The following example provides the *Z* component of a point expressed in spherical coordinates:

**zof([2<45<45])**

The following example provides a point whose *X* and *Y* coordinate values are taken from point *a* and the *Z* coordinate value from point *b:*

**xyof(a)+zof(b)**

# Calculate a Point on a Line

## Quick Reference

The **plt** and **pld** functions return a point on a given line. You can specify the position of the point on the line either by its distance from the first point or parametrically by a *t* parameter.

**pld(p1,p2,dist)** Calculates a point on the line passing through points *p1* and *p2*. The parameter *dist* defines the distance of the point from the point *p1*.

**plt(p1,p2,t)** Calculates a point on the line passing through points *p1* and *p2*. The parameter *t* defines the parametric position of the point on the line.

The following are examples of the parameter *t:*

**If t=0 the point is p1**

**If t=0.5 the point is the midpoint between p1 and p2**

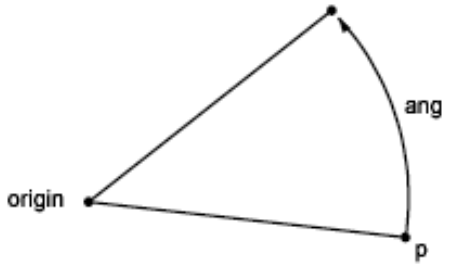
**If t=1 the point is p2**

# Rotate a Point About an Axis

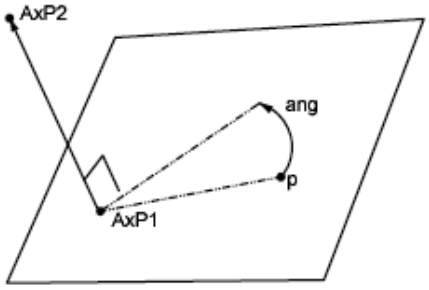
## Quick Reference

The **rot** function rotates a point about an axis and returns the resulting point.

**rot(p,origin,ang)** Rotates point *p* through angle *ang* about the *Z* axis passing through the point *origin,* as shown in the following example:




**rot(p,AxP1,AxP2,ang)** Rotates point *p* through an angle *ang* about the axis passing through points *AxP1* and *AxP2*, as shown in the following example. The axis is oriented from the first point to the second point.



## Obtain an Intersection Point

### Quick Reference

The **ill** and **ilp** functions determine intersection points.

**ill(p1,p2,p3,p4)** Determines the intersection point between two lines (*p1,p2*) and (*p3,p4*). All points are considered three-dimensional.

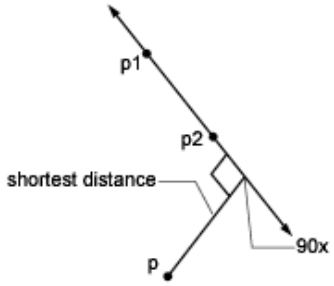
**ilp(p1,p2,p3,p4,p5)** Determines the intersection point between a line (*p1,p2*) and a plane passing through three points (*p3,p4,p5*).

## Calculate a Distance

### Quick Reference

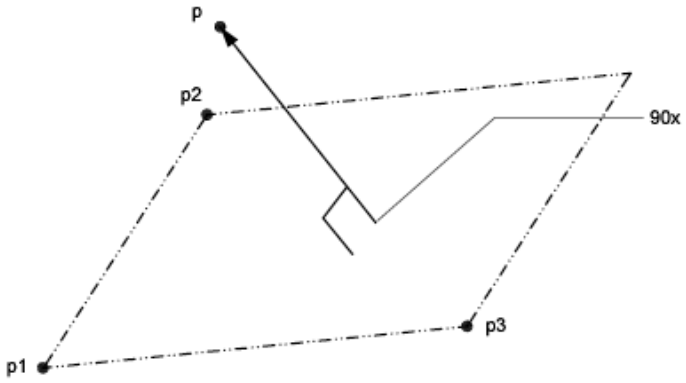
**dist(p1,p2)** Determines the distance between two points, *p1* and *p2*. This is the same as the vector expression **abs(p1-p2)**.

**dpl(p,p1,p2)** Determines the shortest distance between point *p* and the line passing through points *p1* and *p2*.




**dpp(p,p1,p2,p3)** Determines the distance from a point *p* to a plane defined by three points (*p1,p2,p3*).

**dist(p1,p2)** Determines the distance between two points, *p1* and *p2*. This is the same as the vector expression **abs(p1-p2)**.




The following example returns half the distance between the centers of two selected objects:

**dist(cen,cen)/2**

The following example finds the distance between the point 3,2,4 and a plane you define by selecting three endpoints:

**dpp([3,2,4],end, end, end)**

# Obtain a Radius

## Quick Reference

The **rad** function determines the radius of a selected object.

**rad** Determines the radius of a selected object. The object can be a circle, an arc, or a 2D polyline arc segment.

The following example uses **rad** with the *CIRCLE* command. The radius of the new circle is two-thirds of the radius of the selected polyline arc segment:

Command: **circle**

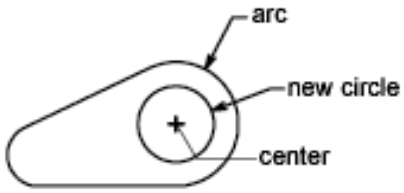
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: **cen**

of *Select the circle*

Specify radius of circle or [Diameter] <*last*>: '**cal**

>> Expression: **2/3*rad**

>> Select circle, arc or polyline segment for RAD function: *Select the circle*




# Obtain an Angle

## Quick Reference

The **ang** function determines the angle between two lines. Angles are measured counterclockwise with respect to either the *X* axis, in the two-dimensional case, or to a user-specified axis, in the three-dimensional case.

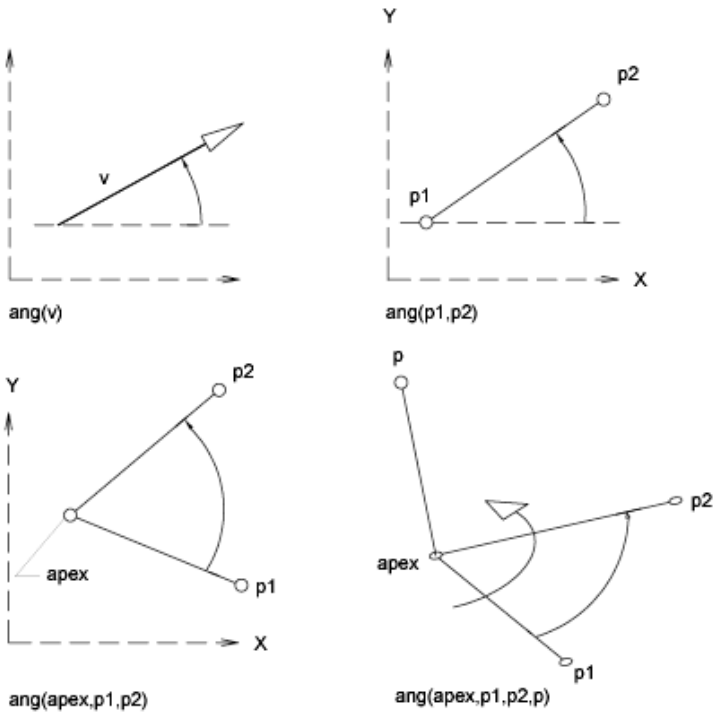
**ang(v)** Determines the angle between the *X* axis and vector *v*. The vector *v* is considered 2D, projected on the *XY* plane of the current UCS.

**ang(p1,p2)** Determines the angle between the *X* axis and the line (*p1*,*p2*), oriented from *p1* to *p2*. The points are considered 2D, projected on the *XY* plane of the current UCS.

**ang(apex,p1,p2)** Determines the angle between lines (*apex,p1*) and (*apex,p2*). The points are considered 2D, projected on the *XY* plane of the current UCS.

**ang(apex,p1,p2,p)** Determines the angle between lines (*apex,p1*) and (*apex,p2*). The lines are considered 3D. The last parameter, point *p*, is used to define the orientation of the angle. The angle is measured counterclockwise with respect to the axis going from apex to *p*.

The following examples show how angles are measured.




You can determine the angle between the two sides of a triangle using the **ang** function, as shown in the following example:

Command: **cal**
>> Expression: **ang(end,end,end)**

Select the apex of the angle, and then select the two opposite vertices.

# Calculate a Normal Vector

## Quick Reference

The **nor** function calculates the unit normal vector (a vector perpendicular to a line or plane), not a point. The vector defines the direction of the normal, not a location in space. You can add this normal vector to a point to obtain another point.

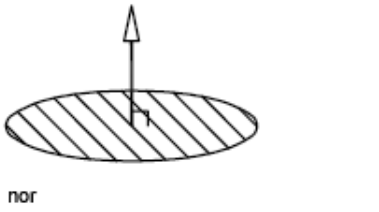
**nor** Determines the three-dimensional unit normal vector of a selected circle, arc, or polyline arc segment. This normal vector is the *Z* coordinate of the object coordinate system (OCS) of the selected object.

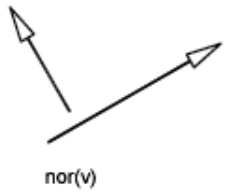
**nor(v)** Determines the two-dimensional unit normal vector to vector *v*. Both vectors are considered 2D, projected on the *XY* plane of the current UCS. The orientation of the resulting normal vector points to the left of the original vector *v*.

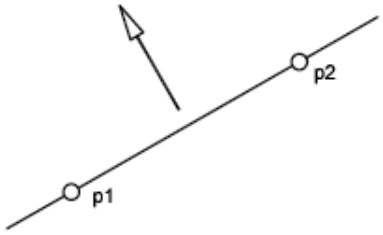
**nor(p1,p2)** Determines the 2D unit normal vector to line *p1,p2*. The line is oriented from *p1* to *p2*. The orientation of the resulting normal vector points to the left from the original line (*p1,p2*).

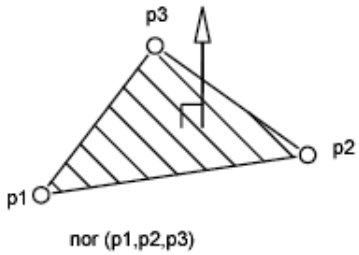
**nor(p1,p2,p3)** Determines the 3D unit normal vector to a plane defined by the three points *p1, p2*, and *p3*. The orientation of the normal vector is such that the given points go counterclockwise with respect to the normal.

The following illustrations show how normal vectors are calculated:




The following example sets the view direction perpendicular to a selected object. The program displays the object in plan view and does not distort the object by the parallel projection.

Command: **vpoint**

Current view direction: VIEWDIR=*current*
Specify a view point or [Rotate] <display compass and tripod>: '**cal**
>> Expression: **nor**
>> Select circle, arc or polyline for NOR function:

# Use Shortcut Functions

## Quick Reference

The functions in the table are shortcuts for commonly used expressions that combine a function with the Endpoint Snap mode.

**Shortcut functions**

| Function | Shortcut for | Description |
|---|---|---|
| dee | dist(end,end) | Distance between two endpoints |
| ille | ill(end,end,end,end) | Intersection of two lines defined by four endpoints |
| mee | (end+end)/2 | Midpoint between two endpoints |
| nee | nor(end,end) | Unit vector in the *XY* plane and normal to two endpoints |
| pldee (d) | pld(d,end,end) | Point at a distance along a line determined by two endpoints (see pld) |
| pltee (t) | plt(t,end,end) | Point at a parametric location on a line determined by two endpoints (see plt) |
| vee | vec(end,end) | Vector from two endpoints |
| vee1 | vec1(end,end) | Unit vector from two endpoints |

# CAMERA

## Quick Reference

Sets a camera and target location to create and save a 3D perspective view of objects

View 

**camera**
Current camera settings: Height=<*current*> Lens Length=<*current*>
Specify camera location <*current*>: *Enter a value or specify a point*

Sets the point from which you view objects in a model.

Specify target location: *Enter a value or specify a point*

Sets the target location of a camera's lens.

Enter an option[? (page 232)/Name (page 232)/LOcation (page 232)/Height (page 232)/Target (page 233)/LEns (page 233)/Clipping (page 233)/View (page 233)/<eXit> (page 233)]:

### ?—List Cameras

Displays a list of the currently defined cameras.

Enter camera name(s) to list <*>: *Enter a name list or press Enter to list all cameras.*

### Name

Names a camera.

Enter name for new camera <Camera1>:

### Location

Specifies the location of the camera.

Specify camera location <current>:

### Height

Changes the camera height.

Specify camera height <current>:

#### Target

Specifies the target of the camera.

Specify camera target <current>:

#### Lens

Changes the lens length of the camera.

Specify lens length in mm <current>:

#### Clipping

Defines front and back clipping planes and sets their values.

Enable front clipping plane? [Yes/No] <No>: *Specify Yes to enable front clipping*
Specify front clipping plane offset from target plane <current>: *Enter a distance*
Enable back clipping Plane? <Yes/No] <No>: *Specify Yes to enable back clipping*
Specify back clipping plane offset from target plane <current>: *Enter a distance*

#### View

Sets the current view to match the camera settings.

Switch to camera view? [Yes/No] <No>:

#### Exit

Cancels the command.

## Camera Preview Dialog Box

### Quick Reference

Displays a preview of a camera view.

**Preview** Displays a preview of a camera view defined with the *CAMERA* command.

**Visual Style** Specifies the visual style applied to the preview.

**Display This Window When Editing a Camera** Specifies that the Camera Preview dialog box is displayed when you edit a camera.

# CHAMFER

## Quick Reference

Bevels the edges of objects

Modify 
Modify ➤ ChamferAt the Command prompt, enter chamfer.
**chamfer**
(TRIM mode) Current chamfer Dist1 = *current*, Dist2 = *current*
Select first line (page 234) or [Undo (page 236)/Polyline (page 236)/Distance (page 236)/Angle (page 237)/Trim (page 237)/mEthod (page 238)/Multiple (page 238)]:
*Use an object selection method or enter an option*

### First Line

Specifies the first of two edges required to define a 2D chamfer, or the edge of a 3D solid to chamfer.

Select second line or shift-select to apply corner: *Use an object selection method or hold down SHIFT and select an object to create a sharp corner*

If you select lines or polylines, their lengths adjust to accommodate the chamfer line. You can hold down SHIFT while selecting the objects to override the current chamfer distances with a value of 0.

If the selected objects are line segments of a 2D polyline, they must be adjacent or separated by no more than one segment. If they're separated by another polyline segment, CHAMFER deletes the segment that separates them and replaces it with the chamfer.

If you select an edge on a 3D solid, you must indicate which one of the two surfaces adjacent to the edge is the base surface.

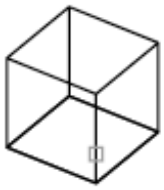
Base surface selection...
Enter surface selection option [Next/OK (current)] <OK>: *Enter* **n** *or* **o***, or press* ENTER

Entering **o** or pressing ENTER sets the selected surface as the base surface.
Entering **n** selects either of the two surfaces adjacent to the selected edge.

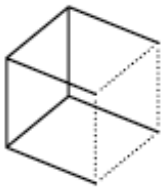
Specify base surface chamfer distance <*current*>:
Specify other surface chamfer distance <*current*>:

After you select the base surface and the chamfer distances, select the edges of the base surface to chamfer. You can select edges individually or all the edges at once.

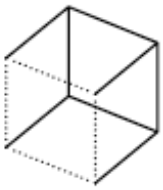
Select an edge or [Loop]: S*elect an edge, enter* **L**, *or press ENTER*



first edge selected



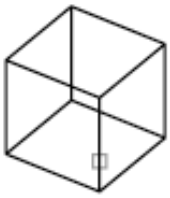
first base surface



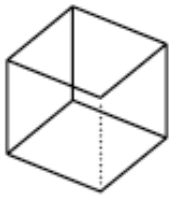
second base surface

### Edge

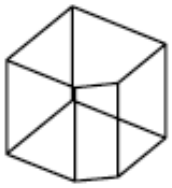
Selects an individual edge to chamfer.



select edge



edge selected

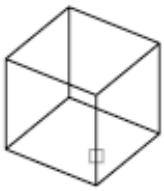


chamfered edge

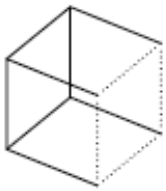
### Loop

Switches to Edge Loop mode.

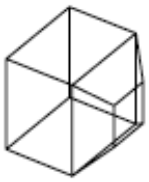
Select an edge loop or [Edge]: *Select an edge, enter* **e**, *or press ENTER*
**Edge Loop** Selects all edges on the base surface.



selecting edge loop



edge loop selected



chamfered edge loop

**Edge** Switches to Edge mode.

### Undo

Reverses the previous action in the command.

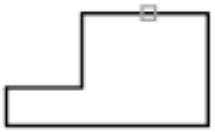
### Polyline

Chamfers an entire 2D polyline.

Select 2D polyline:

The intersecting polyline segments are chamfered at each vertex of the polyline. Chamfers become new segments of the polyline.

If the polyline includes segments that are too short to accommodate the chamfer distance, those segments are not chamfered.



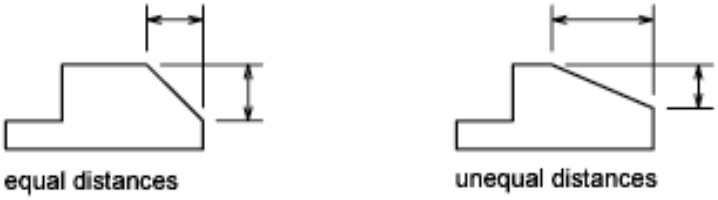
selected polyline

result

### Distance

Sets the distance of the chamfer from the endpoint of the selected edge.

Specify first chamfer distance <*current*>:
Specify second chamfer distance <*current*>:

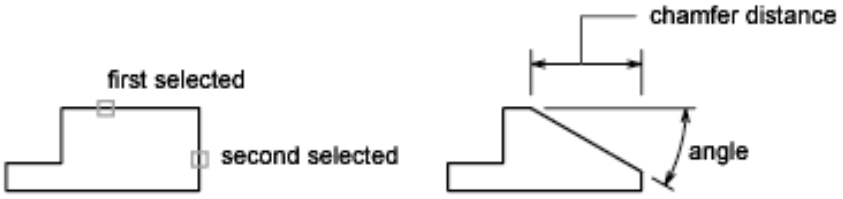



If you set both distances to zero, CHAMFER extends or trims the two lines so they end at the same point.

### Angle

Sets the chamfer distances using a chamfer distance for the first line and an angle for the second line.

Specify chamfer length on the first line <*current*>:
Specify chamfer angle from the first line <*current*>:




### Trim

Controls whether CHAMFER trims the selected edges to the chamfer line endpoints.

Enter Trim mode option [Trim/No trim] <*current*>:

**NOTE** Trim sets the *TRIMMODE* system variable to 1; No Trim sets TRIMMODE to 0.

If the TRIMMODE system variable is set to 1, CHAMFER trims the intersecting lines to the endpoints of the chamfer line. If the selected lines do not intersect, CHAMFER extends or trims them so that they do. If TRIMMODE is set to 0, the chamfer is created without trimming the selected lines.

#### Method

Controls whether CHAMFER uses two distances or a distance and an angle to create the chamfer.

Enter trim method [Distance/Angle] <*current*>:

#### Multiple

Chamfers the edges of more than one set of objects. CHAMFER displays the main prompt and the Select Second Object prompt repeatedly until you press ENTER to end the command.

# CHANGE

## Quick Reference

Changes the properties of existing objects

**change**
Select objects:

Except for zero-thickness lines, the objects selected must be parallel to the current user coordinate system (UCS).

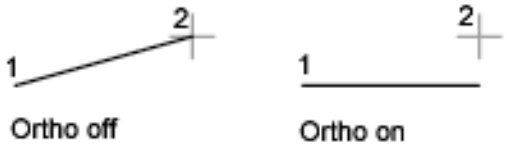
If you select lines and other changeable objects in the same selection set, you get varying results depending on the object selection sequence. The easiest way to use CHANGE is to select only lines in a selection set or select only objects other than lines in a selection set.

Specify change point (page 238) or [Properties (page 240)]: *Specify a new point, or press* ENTER *to enter new values*

#### Change Point or Values

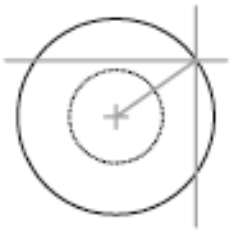
Changes the selected objects. The result depends on the type of objects you select.

**Lines** Moves the endpoints of the selected lines that are closest to the change point to the new point, unless Ortho mode is on. If Ortho mode is on, the selected lines are modified so that they become parallel to either the *X* or the *Y* axis; their endpoints are not moved to the specified coordinate.




**Circles** Changes the circle radius. If you selected more than one circle, the prompt is repeated for the next circle.

Specify new circle radius <no change>:



**Text** Changes text position and other properties.

Specify new text insertion point <no change>:

Specifying a new location repositions the text. Pressing ENTER leaves the text in its original position.

Enter new text style <*current*>:

If the text has a fixed height, the height prompt is not displayed.

Specify new height <*current*>:

Specify new rotation angle <*current*>:

Enter new text <*current*>:

The next object is highlighted and prompts are displayed.

**Attribute Definitions** Changes the text and text properties of an attribute that is not part of a block.

Specify new text insertion point <no change>:

Specifying a new location repositions the text. Pressing ENTER leaves the text at its original position.

Enter new text style <*current*>:

If the text has a fixed height, the height prompt is not displayed

Specify new height <*current*>:

Specify new rotation angle <*current*>:

Enter new tag <*current*>:

Enter new prompt <*current*>:

Enter new default value <*current*>:

**Blocks** Changes the location or rotation of a block.

Specify new block insertion point: *Specify a point (1), or press* ENTER

Specifying a new location repositions the block. Pressing ENTER leaves the block in its original position.

Specify new block rotation angle <*current*>:

The block is rotated about its insertion point to the specified angle.


### Properties

Modifies properties of existing objects.

Enter property to change
[Color/Elev/LAyer/LType/ltScale/LWeight/Thickness/Material/Annotative]:

**NOTE** The Plotstyle option is displayed only when you are using named plot styles.

If you select several objects with different values for the property you want to change, *varies* is displayed as the current value.

You can change several properties at a time. The Enter Property to Change prompt is redisplayed after each option is completed.

### Color

Changes the color of the selected objects.

Enter new color [Truecolor/COlorbook]<*current*>: *Enter a color name or a number from* **1** *through* **255**, *enter* **t**, *enter* **co**, *enter* **bylayer** *or* **byblock**, *or press* ENTER

For example, to change a color to red, enter **red** or **1**. If you enter **bylayer**, the object assumes the color of the layer on which it is located. If you enter **byblock**, the object inherits the color of the block of which it is a component.

**True Color** Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

**Color Book** Specifies a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE**

If you enter a color book name, the following prompt is displayed.

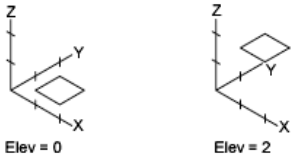
Enter color name: *Enter the name of a color included in the selected color book, such as* **Pantone 573**

### Elev

Changes the *Z*-axis elevation of 2D objects.

Specify new elevation <*current*>:

You can change the elevation of an object only if all its points have the same *Z* value.




### Layer

Changes the layer of the selected objects.

Enter new layer name <*current*>:

### Ltype

Changes the linetype of the selected objects.

Enter new linetype name <*current*>:

If the new linetype is not loaded, the program tries to load it from the standard linetype library file, *acad.lin*. If this procedure fails, use *LINETYPE* to load the linetype.

### Ltscale

Changes the linetype scale factor of the selected objects.

Specify new linetype scale <*current*>:

#### Lweight

Changes the lineweight of the selected objects. Lineweight values are predefined values. If you enter a value that is not a predefined value, the closest predefined lineweight is assigned to the selected objects.

Enter new lineweight <*current*>:

#### Thickness

Changes the *Z*-direction thickness of 2D objects.

Specify new thickness <*current*>:

Changing the thickness of a 3D polyline, dimension, or layout viewport object has no effect.


#### Material

Changes the material of the selected objects if a material is attached.

Enter new material name <ByLayer>:

#### Annotative

Changes the property of the selected objects.

Make annotative? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

# CHECKSTANDARDS

## Quick Reference

Checks the current drawing for standards violations

CAD Standards 

Tools ➤ CAD Standards ➤ CheckAt the Command prompt, enter checkstandards.
**checkstandards**

The Check Standards dialog box (page 243) is displayed.

# Check Standards Dialog Box

## Quick Reference

CAD Standards 
Tools ➤ CAD Standards ➤ CheckAt the Command prompt, enter checkstandards.
**checkstandards**

Analyzes the current drawing for standards violations.

**Problem** Provides a description of a nonstandard object in the current drawing. To fix a problem, select a replacement from the Replace With list, and then click Fix.

**Replace With** Lists possible replacements for the current standards violation. If a recommended fix is available, it is preceded by a check mark. If a recommended fix is not available, no items are highlighted in the Replace With list.

**Preview of Changes** Indicates the properties of the nonstandard object that will be changed if the fix currently selected in the Replace With list is applied.

**Fix** Fixes the nonstandard object using the item currently selected in the Replace With list, and advances to the next nonstandard object in the current drawing. This button is unavailable if a recommended fix does not exist or if an item is not highlighted in the Replace With list.

**Next Problem** Advances to the next nonstandard object in the current drawing without applying a fix.

**Mark This Problem as Ignored** Flags the current problem as ignored. If the Show Ignored Problems option is turned off in the CAD Standards Settings dialog box, problems flagged as ignored are not displayed the next time the drawing is checked.

**Settings** Displays the CAD Standards Settings dialog box (page 1362), which specifies additional settings for the Check Standards dialog box and the Configure Standards dialog box.

**Close** Closes the Check Standards dialog box without applying a fix to the standards violation currently displayed in Problem.

# CHPROP

## Quick Reference

Changes the properties of an object

**chprop**
Select objects:
Enter property to change [Color (page 244)/LAyer (page 245)/LType (page 245)/ltScale (page 245)/LWeight (page 245)/Thickness (page 245)/Material (page 246)/PLotstyle (page 246)/Annotative (page 246)]:

**NOTE** The Plotstyle option is displayed only when you are using named plot styles.

If you select several objects with different values for the property you want to change, *varies* is displayed as the current value.

### Color

Changes the color of the selected objects.

Enter new color [Truecolor/COlorbook]<*current*>: *Enter a color name or a number from* **1** *through* **255**, *enter* **t**, *enter* **co**, *enter* **bylayer** *or* **byblock**, *or press* ENTER

For example, to change a color to red, enter **red** or **1**. If you enter **bylayer**, the object assumes the color of the layer on which it is located. If you enter **byblock**, the object inherits the color of the block of which it is a component.

**True Color** Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

**Color Book** Specifies a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE®**

If you enter a color book name, the following prompt is displayed.

Enter color name: *Enter the name of a color included in the selected color book, such as* **PANTONE® 573**

### Layer

Changes the layer of the selected objects.

Enter new layer name <*current*>:

### Ltype

Changes the linetype of the selected objects.

Enter new linetype name <*current*>:

If the new linetype is not loaded, the program tries to load it from the standard linetype library file, *acad.lin*. If this procedure fails, use *LINETYPE* to load the linetype.

### Ltscale

Changes the linetype scale factor of the selected objects.

Specify new linetype scale <*current*>:

### Lweight

Changes the lineweight of the selected objects. Lineweight values are predefined values. If you enter a value that is not a predefined value, the closest predefined lineweight is assigned to the selected objects.

Enter new lineweight <*current*>:

### Thickness

Changes the *Z*-direction thickness of 2D objects.

Specify new thickness <*current*>:

Changing the thickness of a 3D polyline, dimension, or layout viewport object has no effect.

#### Material

Changes the material of the selected objects if a material is attached.

Enter new material name <ByLayer>:

#### Plotstyle

Changes the plot style of the selected objects. A plot style is a collection of property settings that is saved in a plot style table. This option is displayed only when you are using named plot styles.

Enter Plot Style or [?] <*current*>: *Enter a name or enter* **?**
**Plot Style** Assigns the plot style to the selected objects. When you plot the drawing, objects are plotted with the properties defined in the designated plot style. Plot styles assigned to objects can be overridden by other plot style assignments.

**?—List Plot Styles** Lists all the plot styles defined in the drawing.

#### Annotative

Changes the property of the selected objects.

Make annotative? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

# CHSPACE

## Quick Reference

Moves objects from model space to paper space, or vice versa

Modify ➤ Change SpaceAt the Command prompt, enter chspace.
**chspace**
Select objects:
Set the SOURCE viewport active and press ENTER to continue:

Returns prompts similar to the following:

*N* object(s) changed from MODEL space to PAPER space.
Objects were scaled by a factor of *n* to maintain visual appearance.

The moved object is scaled appropriately in the new space.

# CIRCLE

## Quick Reference

Creates a circle

Draw 

Draw ➤ CircleDoes not exist in the menus.

**circle**

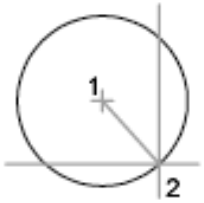
Specify center point (page 247) for circle or [3P (page 248)/2P (page 248)/Ttr (tan tan radius) (page 248)]: *Specify a point or enter an option*

### Center Point

Draws a circle based on a center point and a diameter or a radius.

Specify radius of circle or [Diameter]: *Specify a point, enter a value, enter* **d**, *or press* ENTER

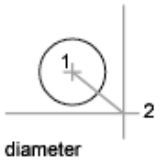
**Radius** Defines the radius of the circle. Enter a value, or specify a point (2). The distance between this point and the center point determines the radius of the circle.




radius

**Diameter** Draws a circle using the center point and a specified distance for the diameter.

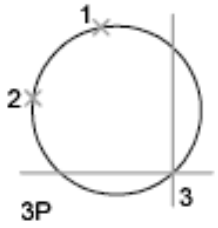
Specify diameter of circle <*current*>: *Specify a point (2), enter a value, or press* ENTER




### 3P (Three Points)

Draws a circle based on three points on the circumference.

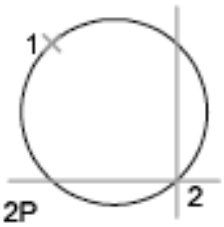
Specify first point on circle: *Specify a point (1)*
Specify second point on circle: *Specify a point (2)*
Specify third point on circle: *Specify a point (3)*




### 2P (Two Points)

Draws a circle based on two endpoints of the diameter.

Specify first endpoint of circle's diameter: *Specify a point (1)*
Specify second endpoint of circle's diameter: *Specify a point (2)*

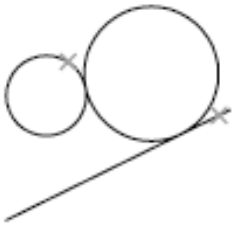



### TTR (Tangent, Tangent, Radius)

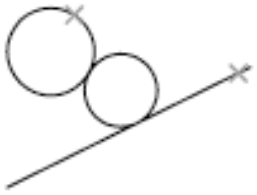
Draws a circle with a specified radius tangent to two objects.

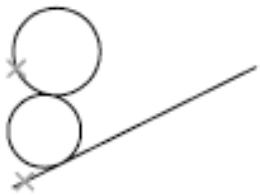
Specify point on object for first tangent of circle: *Select a circle, arc, or line*

Specify point on object for second tangent of circle: *Select a circle, arc, or line*
Specify radius of circle <*current*>:




Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.





# CLEANSCREENON

## Quick Reference

Clears the screen of toolbars and dockable windows (excluding the command line)

View ➤ Clean ScreenDoes not exist in the menus.
**cleanscreenon**

The screen displays only the menu bar, the Model and layout tabs at the bottom of the drawing, the status bar, and the command line. Use *CLEANSCREENOFF* to restore display of interface items except menu bar, status bar, and the command line. Press CTRL+0 (zero) to switch between CLEANSCREENON and CLEANSCREENOFF. A clean screen button is available in the lower-right corner of the application window.

# CLEANSCREENOFF

## Quick Reference

Restores display of toolbars and dockable windows (excluding the command line)

View ➤ Clean ScreenDoes not exist in the menus.
**cleanscreenoff**

Restores the state of the display before *CLEANSCREENON* was used. Use CLEANSCREENON to clear the screen of toolbars and dockable windows (excluding the command line). Press CTRL+0 (zero) to switch between CLEANSCREENON and CLEANSCREENOFF. A clean screen button is available in the lower-right corner of the application window.

# CLOSE

## Quick Reference

Closes the current drawing

File ➤ CloseAt the Command prompt, enter close.
**close**

The current drawing is closed. If you modified the drawing since it was last saved, you are prompted to save or discard the changes.

You can close a file that has been opened in read-only mode if you have made no changes or if you are willing to discard changes. To save changes to a read-only file, you must use the *SAVEAS* command.

# CLOSEALL

## Quick Reference

Closes all currently open drawings

Window ➤ Close AllAt the Command prompt, enter closeall.
**closeall**

All open drawings are closed. A message box is displayed for each unsaved drawing, in which you can save any changes to the drawing before closing it.

# COLOR

### Quick Reference

Sets the color for new objects

Format ➤ ColorAt the Command prompt, enter color.
**color** (or '**color** for transparent use)

The Select Color dialog box (page 251) is displayed.

If you enter **-color** at the command prompt, options are displayed at the command prompt (page 256).

## Select Color Dialog Box

### Quick Reference

Format ➤ Color
**color** (or '**color** for transparent use)

Define the color of objects. You can select from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

- Index Color (page 251)
- True Color (page 253)
- Color Books (page 255)

## Index Color Tab (Select Color Dialog Box)

### Quick Reference

Specifies color settings using the 255 AutoCAD Color Index (ACI) colors.

### AutoCAD Color Index (ACI) Palettes

Specifies a color from the AutoCAD Color Index. If you hover over a color, the number of the color and its red, green, blue value are displayed below the palette. Click a color to select it, or enter the color number or name in the Color box.

The large palette displays colors 10 through 249.

The second palette displays colors 1 through 9; these colors have names as well as numbers.

The third palette displays colors 250 through 255; these colors are shades of gray.

### Index Color

Indicates the ACI color number when you hover over a color.

### Red, Green, Blue

Indicates the RGB color value when you hover over a color.

### Bylayer

Specifies that new objects assume the color assigned to the layer on which you create them. When BYLAYER is selected, the color of the current layer is displayed in the Old and New color swatches.

### Byblock

Specifies that new objects use the default color (white or black, depending on your background color) until you group the objects into a block and insert the block. When you insert the block into a drawing, the objects in the block inherit the current Color setting.

---

**NOTE** The BYLAYER and BYBLOCK options do not apply to the *LIGHT* command.

---

### Color

Specifies a color name, BYLAYER or BYBLOCK color, or an AutoCAD Color Index (ACI) number of 1 through 255. The New color swatch shows the most recently selected color.

#### Old Color Swatch

Displays the previously selected color.

#### New Color Swatch

Displays the currently selected color.

## True Color Tab (Select Color Dialog Box)

### Quick Reference

Specifies color settings using true colors (24-bit color) with either the Hue, Saturation, and Luminance (HSL) color model or the Red, Green, and Blue (RGB) color model. Over sixteen million colors are available when using true color functionality. The options available on the True Color tab are dependent on whether the HSL or RGB color model is specified.

#### HSL Color Model

Specifies the HSL color model for selecting colors.

Hue, saturation, and luminance are properties of colors. By manipulating the values of these properties, you can specify a wide range of colors.

**Hue** Specifies the hue of a color. Hues represent a specific wavelength of light within the visible spectrum. To specify a hue, use the color spectrum or specify a value in the Hue box. Adjusting this value affects the RGB value. Valid hue values are from 0 to 360 degrees.

**Saturation** Specifies the purity of a color. High saturation causes a color to look more pure while low saturation causes a color to look washed-out. To specify color saturation, use the color spectrum or specify a value in the Saturation box. Adjusting this value affects the RGB value. Valid saturation values are from 0 to 100%.

**Luminance** Specifies the brightness of a color. To specify color luminance, use the color slider or specify a value in the Luminance box. Valid luminance values are from 0 to 100%. A value of 0% represents the color black, 100% represents white, and 50% represents the optimal brightness for the color. Adjusting this value also affects the RGB value.

**Color Spectrum** Specifies the hue and purity of a color. To specify a hue, move the crosshairs from side to side over the color spectrum. To specify color saturation, move the crosshairs from top to bottom over the color spectrum.

**Color Slider** Specifies the brightness of a color. To specify color luminance, adjust the bar on the color slider or specify a value in the Luminance box.

### RGB Color Model

Specifies the RGB color model for selecting colors. The options available on the True Color tab are dependent on whether the HSL or RGB color model is specified.

Colors can be broken down into components of red, green, and blue. The values specified for each component represent the intensity of the red, green, and blue components. The combination of these values can be manipulated to create a wide range of colors.

**Red** Specifies the red component of a color. Adjust the slider on the color bar or specify a value from 1 to 255 in the Red box. If this value is adjusted, it will be reflected in the HSL color mode values.

**Green** Specifies the green component of a color. Adjust the slider on the color bar or specify a value from 1 to 255 in the Green box. If this value is adjusted, it will be reflected in the HSL color mode values.

**Blue** Specifies the blue component of a color. Adjust the slider on the color bar or specify a value from 1 to 255 in the Blue box. If this value is adjusted, it will be reflected in the HSL color mode values.

### Color

Specifies the RGB color value. This option is updated when changes are made to HSL or RGB options. You can also edit the RGB value directly using the following format: **000,000,000**.

### True Color Stored as RGB

Indicates the value for each RGB color component.

### Old Color Swatch

Displays the previously selected color.

#### New Color Swatch

Displays the currently selected color.

## Color Books Tab (Select Color Dialog Box)

### Quick Reference

Specifies colors using third-party color books (such as PANTONE®) or user-defined color books. Once a color book is selected, the Color Books tab displays the name of the selected color book.

#### Color Book

Specifies the color book to be used when selecting colors. The list consists of all the color books that are found in the Color Book Locations specified in the Options dialog box, Files tab.

Displays the pages of the selected color book and the colors and color names on each page. Color books containing up to ten colors per page are supported. If a color book is not paginated, the colors are organized into pages containing seven colors per page. To view color book pages, select an area on the color slider or use the up and down arrows to browse.

#### RGB Equivalent

Indicates the value for each RGB color component.

#### Color

Indicates the currently selected color book color. You can search for a specific color in a color book by entering the number of the color swatch and pressing TAB. This action updates the New color swatch with the requested color number. If the specified color is not found in the color book, the closest number match is displayed.

#### Old Color Swatch

Displays the previously selected color.

#### New Color Swatch

Displays the currently selected color.

# -COLOR

## Quick Reference

If you enter **-color** at the command prompt, the following COLOR command prompts are displayed.

Enter default object color [Truecolor (page 256)/COlorbook (page 256)]<BYLAYER>: *Enter a color, enter***t***, enter***co***, or press* ENTER

You can enter a color from the AutoCAD Color Index (a color name or number), a true color, or a color from a color book.

You can enter the color number (1 through 255) or the color name (the names for the first seven colors). For example, you can specify the color red by entering the ACI number **1** or the ACI name **red**.

You can also enter **bylayer** or **byblock**. If you enter **byblock**, all new objects are drawn in the default color (white or black, depending on your background color) until they are grouped into a block. When you insert the block in a drawing, the objects in the block inherit the current setting of COLOR.

---

**WARNING** If you used a mixture of color methods to draw the objects that make up a block, inserting that block or changing its color produces complex results.

---

If you enter **bylayer**, new objects assume the color assigned to the layer on which you create them. See the *LAYER* command for information about assigning a color to a layer.

#### True Color

Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

#### Color Book

Specifies a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* Pantone

If you enter a color book name, you are prompted to enter the color name in the color book.

Enter color name: *Enter the name of a color included in the selected color book, such as* Pantone 573

# COMMANDLINE

## Quick Reference

Displays the command line

Tools ➤ Command LineDoes not exist in the menus.
**commandline**

Displays the command window when it has been hidden.

# COMMANDLINEHIDE

## Quick Reference

Hides the command line

Tools ➤ Command LineDoes not exist in the menus.
**commandlinehide**

Hides the command window. When the command window is hidden, you can still enter commands with dynamic prompts turned on.

# COMPILE

## Quick Reference

Compiles shape files and PostScript font files into SHX files

**compile**

The Select Shape or Font File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the SHP or PFB file name in the dialog box. The compiled file is assigned this name with the file extension *.shx*.

# CONE

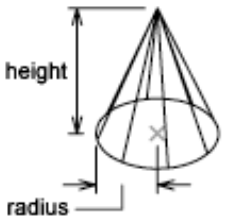
## Quick Reference

Creates a 3D solid with a circular or elliptical base tapering symmetrically to a point or a circular or elliptical planar face

Modeling 
Draw ➤ Modeling ➤ ConeAt the Command prompt, enter cone.
**cone**
3D Make panel, Cone
Specify center point of base or [3P (page 259)/2P (page 260)/Ttr (page 260)/Elliptical (page 261)]: *Specify a point (1) or enter an option*
Specify base radius or [Diameter (page 262)] <default>: *Specify a base radius, enter* **d** *to specify a diameter, or press* ENTER *to specify the default base radius value*
Specify height or [2Point (page 263)/Axis endpoint (page 263)/Top radius (page 264)] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Use the Top Radius option to create a cone frustum.

Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.




### 3P (Three Points)

Defines the base circumference and base plane of the cone by specifying three points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*
Specify third point: *Specify a point*
Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

**Top Radius** Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

### 2P (Two Points)

Defines the base diameter of the cone by specifying two points.

Specify first endpoint of diameter: *Specify a point*
Specify second endpoint of diameter: *Specify a point*
Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

**Top Radius** Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

### TTR (Tangent, Tangent, Radius)

Defines the base of the cone with a specified radius tangent to two objects.

Specify point on object for first tangent: *Select a point on an object*
Specify point on object for second tangent: *Select a point on an object*
Specify radius of circle <default>: *Specify a base radius or press* ENTER *to specify the default base radius value*

Sometimes, more than one base matches the specified criteria. The program draws the base of the specified radius whose tangent points are closest to the selected points.

Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

**Top Radius** Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

### Elliptical

Specifies an elliptical base for the cone.

Specify endpoint of first axis or [Center]: *Specify a point*
Specify other endpoint of first axis: *Specify a point*
Specify endpoint of second axis: *Specify a point*
Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**Center** Creates the base of the cone by using a specified center point.

Specify center point: *Specify a point*

Specify distance to first axis <default>: *Specify a distance or press* ENTER *to specify the default distance value*

Specify endpoint of second axis: *Specify a point*

Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

**Top Radius** Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

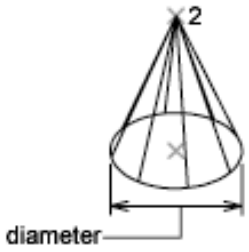
Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

### Diameter

Specifies the diameter for the base of the cone.

Specify diameter <default>: *Specify a diameter or press* ENTER *to specify the default value*

Initially, the default diameter is not set to any value. During a drawing session, the default value for the diameter is always the previously entered diameter value for any solid primitive.




**2Point** Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

**Top Radius** Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

### 2Point

Specifies that the height of the cone is the distance between the two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

### Axis Endpoint

Specifies the endpoint location for the cone axis. The axis endpoint is the top point of the cone or the center point of the top face of the cone frustum (Top Radius option). The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cone.

Specify axis endpoint: *Specify a point*

#### Top Radius

Specifies the top radius of the cone, creating a cone frustum.

Specify top radius <default>: *Specify a value or press* ENTER *to specify the default value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

# CONVERT

## Quick Reference

Optimizes 2D polylines and associative hatches created in AutoCAD Release 13 or earlier

**convert**
Enter type of objects to convert [Hatch (page 264)/Polyline (page 264)/All (page 264)] <All>: *Enter* **h** *for hatches,* **p** *for polylines, or* **a** *for both*
**Hatch** Converts all hatches in the drawing.

Enter object selection preference [Select/All] <All>: *Enter* **s** *to select specific objects to convert or* **a** *to convert all objects in the drawing*

**Polyline** Converts all polylines in the drawing.

Enter object selection preference [Select/All] <All>: *Enter* **s** *to select specific objects to convert or* **a** *to convert all objects in the drawing*

**All** Converts all polylines and hatches in the drawing.

Enter object selection preference [Select/All] <All>: *Enter* **s** *to select specific objects to convert or* **a** *to convert all objects in the drawing*

One or both of the following messages are displayed:

*number* hatch objects converted
*number* 2d polyline objects converted

Hatches are not updated automatically when a drawing from a previous release is opened in Release 14 or later. Information about the rotation of a hatch pattern may not be updated properly if you have changed the UCS since

creating the hatch. When updating hatches with CONVERT, it is recommended that you use the Select option so that you can check your results.

In most cases, you do not need to update polylines with CONVERT. By default, the *PLINETYPE* system variable specifies that polylines are updated automatically when you open an older drawing. Polylines may be created in the old format by third-party applications, and they may be contained in an older drawing that was inserted as a block and then exploded.

**NOTE** Polylines containing curve-fit or splined segments always retain the old format, as do polylines that store extended object data on their vertices. Editing commands make no distinction between the two formats.

# CONVERTCTB

## Quick Reference

Converts a color-dependent plot style table (CTB) to a named plot style table (STB)

**convertctb**

Displays the Select File dialog box (a standard file selection dialog box (page 931)), where you can select the color-dependent plot style table file you want to convert. The Save As dialog box is then displayed. If necessary, specify a new location or name for the converted plot style table file.

CONVERTCTB saves a copy of a color-dependent plot style table as a named plot style table, which you can then attach to drawings that use named plot styles. The original color-dependent plot style table is not affected by CONVERTCTB. The default location for the new named plot style table file is in the *Plot Styles* folder. The default name for the new named plot style table file is the same as the color-dependent plot style table name.

CONVERTCTB creates one named plot style for each color that has unique plot properties, one named plot style for each group of colors that are assigned the same plot properties, and a default named plot style called NORMAL.

For example, if all the colors in a color-dependent plot style table have the same plot properties, CONVERTCTB creates only two named plot styles in the named plot style table: NORMAL, the default plot style, and STYLE 1, which assumes the plot properties that were assigned to all the colors in the color-dependent plot style table. However, suppose that all the colors in a color-dependent plot style table have the same plot properties except for two

colors, BLUE and GREEN. If these colors each have unique plot properties, CONVERTCTB creates four named plot styles in the named plot style table: NORMAL, the default plot style; STYLE 1, which assumes the plot properties of GREEN; STYLE 2, which assumes the plot properties of BLUE; and STYLE 3, which assumes the plot properties that were assigned to all the other colors.

CONVERTCTB gives the plot styles in the new named plot style table generic names such as STYLE 1, STYLE 2. You can modify these generic plot style names in the the Plot Style Table Editor. Click the Edit button on the Plot Device tab in the Page Setup dialog box (page 1005). If you want to rename the plot styles, you must do so before applying them to drawing layouts.

# CONVERTOLDLIGHTS

## Quick Reference

Converts lights created in previous releases to lights in AutoCAD 2007 format

convertoldlights

The lights in the drawing are converted to AutoCAD 2007 format.

**WARNING** The conversion may not be correct in all cases. You may need to adjust intensity, for example.

# CONVERTOLDMATERIALS

## Quick Reference

Converts materials created in previous releases to materials in AutoCAD 2007 format

convertoldmaterials

The materials in the drawing are converted to AutoCAD 2007 format.

**WARNING** The conversion may not be correct in all cases. You may need to adjust material mapping, for example.

# CONVERTPSTYLES

## Quick Reference

Converts the current drawing to either named or color-dependent plot styles

**convertpstyles**

A drawing can use either named or color-dependent plot styles, but not both.

CONVERTPSTYLES converts a currently open drawing from color-dependent plot styles to named plot styles, or from named plot styles to color-dependent plot styles, depending on which plot style method the drawing is currently using.

- Converting drawings from color-dependent plot styles to named plot styles (page 267)
- Converting drawings from named plot styles to color-dependent plot styles (page 268)

When you convert a drawing, CONVERTPSTYLES sets the *PSTYLEMODE* system variable to the appropriate setting (1 = named, 0 = color-dependent).

For example, a drawing using color-dependent plot styles assigns plot properties to objects and layers by color. In other words, all objects with the same color have the same plot properties. CONVERTPSTYLES converts the drawing to use named plot styles that can be applied to objects or layers independent of color assignment. In other words, all objects with the same color can have different plot properties.

### Converting Drawings from Color-Dependent to Named Plot Styles

A dialog box advises you to convert the drawing's plot style tables before converting the drawing. You can convert the color-dependent plot style tables assigned to the drawing to named plot style tables using *CONVERTCTB*.

The Select File dialog box (a standard file selection dialog box (page 931)) is displayed, where you select the named plot style table file to attach to the converted drawing.

CONVERTPSTYLES requires you to select a named plot style table that was converted using CONVERTCTB or created from a PC2 or PCP file. Normally you should select the named plot style table that was converted from the color-dependent plot style table that was assigned to the same drawing.

CONVERTPSTYLES attaches the selected named plot style table to model space and to all layouts. Drawing layers are each assigned a named plot style (from the converted plot style table) that has the same plot properties that their color-dependent plot style had. Drawing objects that had the same color-dependent plot style as their layer are assigned the named plot style BYLAYER. Drawing objects that had a color-dependent plot style that was different from their layer are assigned a named plot style that has the same properties that their color-dependent plot style had.

After a drawing is converted to named plot styles, you can change the named plot style table assignment or assign other named plot styles tables to model space or layouts. You can also assign individual plot styles to drawing objects.

#### Converting Drawings from Named to Color-Dependent Plot Styles

A dialog box advises you that the named plot styles attached to objects and named plot style tables attached to model space and layouts will be detached.

After a drawing is converted to color-dependent plot styles, you can assign a color-dependent plot style table. Plot styles will be applied by color.

# CONVTOSOLID

## Quick Reference

Converts polylines and circles with thickness to 3D solids

Modify ➤ 3D Operations ➤ Convert to Solid
**convtosolid**
3D Make panel (click icon to expand), Convert to Solid

With the CONVTOSOLID command, you can convert the following objects into extruded 3D solids:

- Uniform-width wide polylines with thickness
- Closed, zero-width polylines with thickness
- Circles with thickness

**NOTE** You cannot use CONVTOSOLID with polylines that contain vertices with 0 width or that contain segments of variable width.

You can select the objects to convert before you start the command.

Select objects: *Select one or more objects with thickness to convert into extruded 3D solids*

If one or more objects in the selection set are invalid for the command, you will be prompted again to select objects.

The *DELOBJ* system variable controls whether the objects you select are automatically deleted when the solid is created or whether you are prompted to delete the objects.

# CONVTOSURFACE

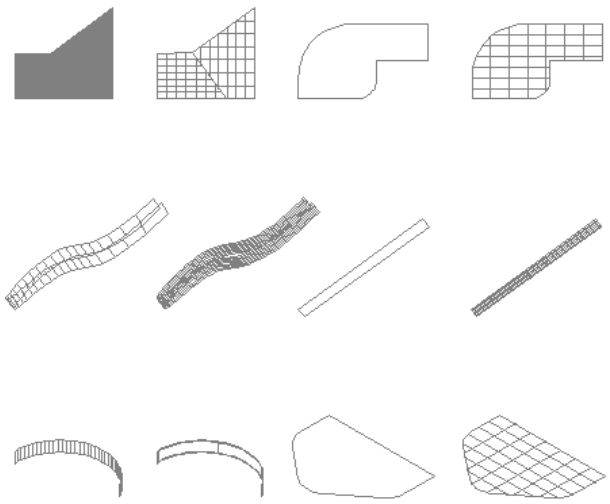
## Quick Reference

Converts objects to surfaces

Modify ➤ 3D Operations ➤ Convert to Surface
**convtosurface**
3D Make panel (click icon to expand), Convert to Surface

With the CONVTOSURFACE command, you can convert the following objects into surfaces:

- 2D solids
- Regions
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness
- Planar 3D faces



You can select the objects to convert before you start the command.

**NOTE** You can create surfaces from 3D solids with curved faces, such as a cylinder, with the *EXPLODE* command.

Select objects: *Select one or more objects to convert into surfaces*

If one or more objects in the selection set are invalid for the command, you will be prompted again to select objects.

The *DELOBJ* system variable controls whether the objects you select are automatically deleted when the surface is created or whether you are prompted to delete the objects.

# COPY

## Quick Reference

Copies objects a specified distance in a specified direction

Modify 

Modify ➤ CopyDoes not exist in the menus.

Select the objects to copy, and right-click in the drawing area. Click Copy Selection.

**copy**

Select objects: *Use an object selection method and press ENTER when you finish*
Current settings: Copy mode = *current*
Specify base point or [Displacement/mOde/Multiple]<Displacement>: *Specify a base point or enter an option*

The two points you specify define a vector that indicates how far the copied objects are to be moved and in what direction.

If you press ENTER at the Specify Second Point prompt, the first point is interpreted as a relative *X,Y,Z* displacement. For example, if you specify **2,3** for the base point and press ENTER at the next prompt, the objects are copied 2 units in the *X* direction and 3 units in the *Y* direction from their current position.

The COPY command repeats automatically by default. To exit the command, press ENTER.

### Displacement

Specifies a relative distance and direction using coordinates.

Specify displacement <*last value*>: *Enter coordinates to represent a vector*

### Mode

Controls whether the command repeats automatically. This setting is controlled by the *COPYMODE* system variable.

Enter a copy mode option [Single/Multiple] <*current*>: *Enter* **s** *or* **m**

### Multiple

Overrides the Single mode setting. The COPY command is set to repeat automatically for the duration of the command. This setting is controlled by the *COPYMODE* system variable.

**NOTE** This option is displayed only when copy mode is set to Single.

# COPYBASE

## Quick Reference

Copies objects with a specified base point

Edit ➤ Copy with Base Point
End any active commands, right-click in the drawing area, and choose Copy with Base Point.
**copybase**
Specify base point:
Select objects:

The selected objects are copied to the Clipboard. Use *PASTECLIP* to move the copied objects from the Clipboard to a location in the same document or to another document. When you paste an object copied with COPYBASE, it is placed relative to the specified base point.

# COPYCLIP

## Quick Reference

Copies objects or command prompt text to the Clipboard

Standard 
Edit ➤ CopyAt the Command prompt, enter copyclip.
End any active commands, right-click in the drawing area, and choose Copy.
**copyclip**
Select objects:

COPYCLIP copies all objects you select to the Clipboard. You can paste the contents of the Clipboard into a document or drawing as an OLE object.

**NOTE** You can also use CTRL+C to run COPYCLIP. If the cursor is in the drawing area, the selected objects are copied to the Clipboard. If the cursor is on the command line or in the text window, the selected text is copied to the Clipboard.

# COPYHIST

## Quick Reference

Copies the text in the command prompt history to the Clipboard

**copyhist**

The text is copied to the Clipboard.

# COPYLINK

## Quick Reference

Copies the current view to the Clipboard for linking to other OLE applications

Edit ➤ Copy LinkAt the Command prompt, enter copylink.
**copylink**

COPYLINK copies the current view to the Clipboard. You can paste the contents of the Clipboard into a document as an OLE object.

# COPYTOLAYER

## Quick Reference

Copies one or more objects to another layer

Layers II 
Format ➤ Layer Tools ➤ Copy Objects to New Layer
**copytolayer**
Select objects to copy:
Select object on destination layer (page 274) or [Name (page 274)] <Name>:
*Select an object or enter* **n**

If you enter **-copytolayer** at the command prompt, options are displayed at the command prompt (page 275).

#### Select Object on Destination Layer

Specifies the layer on which the selected objects are placed.

Specify Base Point or [Displacement/Exit] <eXit>: *Specify a point, enter* **d**, *or enter* **x**
**Specify Base Point** Specifies the base point of the copied objects.
Specify second point of displacement or <use first point as displacement>:

**Displacement** Enters coordinate values that specify a relative distance and direction.
Specify displacement <0.0000, 0.0000, 0.0000>:

**Exit** Cancels the command.

#### Name

Displays the Copy to Layer dialog box (page 274).

Specify Base Point or [Displacement/Exit] <eXit>: *Specify a point, enter* **d**, *or enter* **x**
**Specify Base Point** Specifies the base point of the copied objects.
Specify second point of displacement or <use first point as displacement>:

**Displacement** Enters coordinate values that specify a relative distance and direction.
Specify displacement <0.0000, 0.0000, 0.0000>:

**Exit** Cancels the command.

## Copy To Layer Dialog Box

### Quick Reference

Specifies the layer on which the selected objects are placed.

#### Destination Layer

Displays a list of layers that you can select as the destination layer. You can also enter a name to create a new layer. The new layer inherits the properties (on/off, freeze/thaw, locked/unlocked, etc.) of the current layer.

# -COPYTOLAYER

## Quick Reference

If you enter **-copytolayer** at the command prompt, the following COPYTOLAYER command prompts are displayed.

Select objects to copy:
Specify the destination layer name (page 275) or [? (page 275)/= (page 275) (select object)] <0>: *Select an object on the destination layer, enter***?***, or enter=*

### Specify the Destination Layer Name

Displays a list of layers that you can select as the destination layer. You can also enter a name to create a new layer. The new layer inherits the properties (on/off, freeze/thaw, locked/unlocked, etc.) of the current layer

N object(s) copied and placed on layer "*<layer name>*".

### ?—List Layers

Enter layer name(s) to list <*>: *

### =—Layer By Object

Selects a destination layer by selecting an object on that layer.

Select an object with the desired layer name: *Select an object on the destination layer*

# CUI

## Quick Reference

Manages customized user interface elements such as workspaces, toolbars, menus, shortcut menus and keyboard shortcuts

Tools ➤ Customize ➤ InterfaceAt the Command prompt, enter cui.
**cui**

The Customize User Interface Dialog Box (page 276) is displayed.

**NOTE** The XML-based CUI file, replaces both the Legacy Menu File (MNS) and Legacy Menu Template (MNU) files used in past releases.

For information about customizing the different user interface elements found in the Customize User Interface dialog box, see Customize the User Interface in the Customization Guide.

# Customize User Interface Dialog Box

## Quick Reference

Tools ➤ Customize ➤ InterfaceAt the Command prompt, enter cui.
**cui**

Manages customized user interface elements such as workspaces, toolbars, menus, shortcut menus, and keyboard shortcuts.

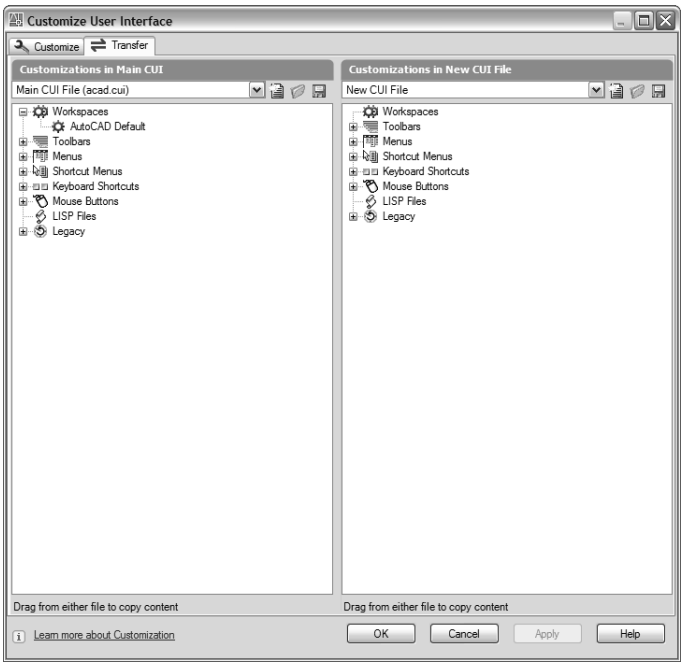
- Transfer tab (Customize User Interface dialog box) (page 276)
- Customize tab (Customize User Interface dialog box) (page 277)

For information about customizing the different user interface elements found in the Customize User Interface dialog box, see Customize the User Interface in the Customization Guide.

# Transfer Tab (Customize User Interface Dialog Box)

## Quick Reference

Transfers user interface elements to or from a main or partial customization (CUI) file, where your interface element data is stored. You open a CUI, MNS, or MNU file to import or export user interface data. When you open MNU or MNS files, they are automatically converted to a CUI file format. The original MNU or MNS files are not modified.




**Customizations In panes** When you enter **cuiimport** at the command prompt, displays the main CUI file (*acad.cui* by default) in the right pane. You open a CUI or MNU file in the left pane to transfer data. When you enter **cuiexport** at the command prompt, displays the main CUI file in the left pane. You open another CUI or MNU file in the right pane to transfer data.

# Customize Tab (Customize User Interface Dialog Box)

## Quick Reference

Provides an interface for customizing workspaces, toolbars, menus, shortcut menus, and other user interface elements.

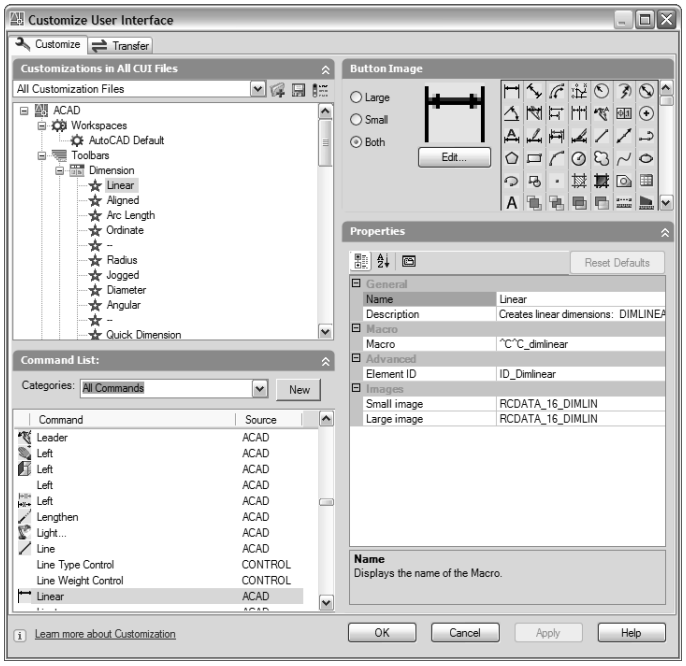
---

**NOTE** When the Customize User Interface dialog box is displayed using the Customize Commands option from the Tool Palettes window, the Customizations In pane is collapsed and the Command List pane is expanded.

---

### Customizations In Pane

Displays a tree structure of user interface elements that can be customized, such as workspaces, toolbars, menus, partial CUI files, and so on.




**List box** Displays a list of CUI files that are loaded and an Open option.

**Available Customizations In Toolbar** Contains Open, Save, and Display Filters buttons.

**Open** Loads a CUI file that you can add to the *acad.cui* file.

**Save** Saves changes to all displayed CUI files.

**Display Filters** Opens the Display Filters dialog box, where you can select user interface elements to display or hide in the tree view.

**Tree View** Lists the CUI file(s) selected in the list box, and displays those interface elements selected in the Display Filters dialog box.

### Dynamic Display Pane

Displays content specific to the user interface element you select in the tree view in the left pane. Following are the types of display content you'll see in

the right pane and the corresponding information displayed in the left pane's tree view:

**Information** Describes selected user interface elements that do not have properties (such as the Toolbars node, Menus node, and Shortcut Menus node).

**Properties** Displays the properties of user interface elements or items selected in the tree view. Elements or items that display properties include specific menus, menu commands, toolbars, and toolbar buttons.

**Button Image** Displays the toolbar buttons of all toolbars loaded in the program. You can edit the toolbar button, change its properties, and create a new toolbar button.

**Shortcuts** Displays a list of shortcut key names and temporary override names, their corresponding shortcut key combinations, keyboard shortcut type (shortcut key or temporary override), and the source CUI file name.

### Command List Pane

Displays a list of commands that are loaded in the program.

**Categories** Displays filters for the command list, including All Commands, ACAD Commands, User Defined, or Control Commands. You can also filter the list to display commands within specific menu.

**New** Creates a new command. You can add or change properties and create or edit a button.

### Properties Pane

Displays user interface properties that you can view, edit, or delete.

**General** Displays the name and description of a user interface element. The name you enter is the label displayed in the program. The description you enter is displayed in the status bar. For workspaces, determines the display of layout and model tabs (On or Off), screen menu display (On or Off), and scroll bar display (On or Off). For image tile menus, determines whether there is a blank line (Yes or No), and defines the slide library and slide name.

**Appearance** For toolbars, determines whether the toolbar is on by default (Hide or Show), the toolbar orientation (Floating,Top, Bottom, Left, Right), and the number of rows the toolbar displays. For screen menus, determines the start line of the screen menu.

**Macro** For toolbar commands, menu commands, mouse buttons, and image tile menus, displays the macro assigned to a selected command. You can create a macro or edit an existing macro.

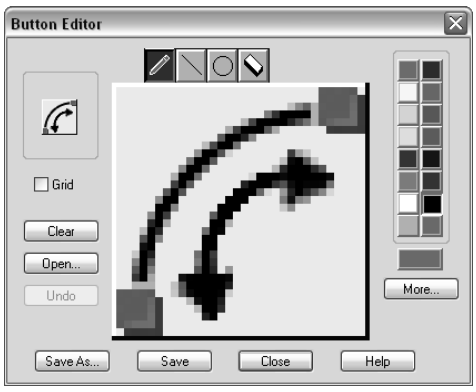
**Advanced** Displays the DIESEL strings, aliases, version compatibility, and element IDs that you can define for each user interface element.

**Images** For toolbar command, menu commands, mouse buttons, and image tile menus, displays a small image and large image box, where you can define an image's resource image (Yes or No) and name (where you browse to your user-defined images).

# Button Editor Dialog Box

## Quick Reference

Tools ➤ Customize ➤ InterfaceAt the Command prompt, enter cui.
Shortcut ➤ Right-click any toolbar and click Customize
**cui**




Modifies or creates buttons.

### Button Image

Displays the button at its actual size.

### Editing Tools

Provides tools for editing a button image.

**Pencil** Edits one pixel at a time using the current color.

**Line** Creates lines using the current color. Click and drag to set the start point and draw the line. Release to complete the line.

**Circle** Creates circles using the current color. Click to set the center and drag to set the radius. Release to complete the circle.

**Erase** Sets pixels to white. Click and drag over colored pixels to change them to white.

### Color Palette

Sets the current palette used by the editing tools.

### More

Opens the Select Color dialog box.

### Editing Area

Provides a close-up view of the button image for editing.

### Grid

Displays a grid in the editing area. Each grid square represents a single pixel.

### Clear

Clears the editing area.

### Open

Opens an existing button image for editing. Button images are stored as bitmap (BMP) files.

### Undo

Undoes the last action.

### Save

Saves the customized button image.

**Save As**

Saves the customized button image using a different name or location.

**NOTE** When saving a new button image, the Create File dialog box will default to the folder defined under Custom Icon Location of the Files tab in the Options dialog box. Button images that are located in this folder can be migrated with the Migrate Custom Settings dialog box in future releases.

# Find and Replace Dialog Box

## Quick Reference

Tools ➤ Customize ➤ InterfaceAt the Command prompt, enter cui.
**cui**

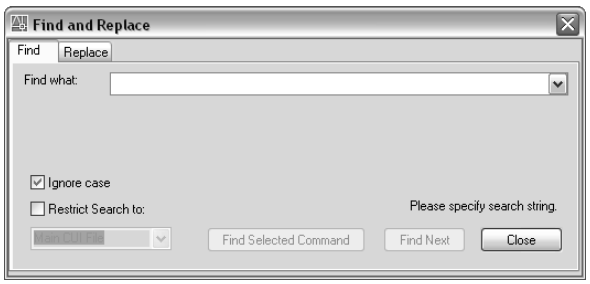
Locates and replaces commands or command properties (command names, descriptions, or macros).

- Find tab (Find and Replace dialog box) (page 282)
- Replace tab (Find and Replace dialog box) (page 283)

# Find Tab (Find and Replace Dialog Box)

## Quick Reference

Searches for commands or command properties in either the Command List pane or the Available Customizations in *<file name>* pane.



**Find What** Displays the search string you want to locate. If you select a command in the Command List pane, this box displays that string. You can also enter a string. Any previous strings entered in this box are stored in the drop-down list.

**Ignore Case** Locates all commands or command properties regardless of their case (for example, the program would search for both LINE and line).

**Restrict Search To** Limits the search to the CUI file you select from the drop-down list. By default, the main CUI file (*acad.cui* by default) is searched.

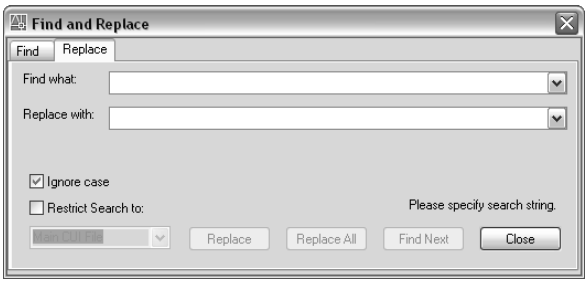
**Find Selected Command** When a command is selected in the Command List pane, locates the next interface element in the tree view that references the selected command.

**Find Next** Locates the next instance of the search string in the Name, Description, or Macros box in the Properties pane. If you search in the Command List pane, only commands in the list are located. If you search in the tree view in the Customizations In pane, the search starts in the tree view and continues to the commands in the Command List pane.

## Replace Tab (Find and Replace Dialog Box)

### Quick Reference

Replaces a search string with a new search string.




**Find What** Displays the search string you want to locate. If you select a command in the Command List pane, this box displays that string. You can also enter a string. Any previous strings entered in this box are stored in the drop-down list.

**Replace With** Displays the string that will replace the string you searched for.

**Ignore Case** Locates all commands or command properties regardless of their case (for example, the program would replace both LINE and line).

**Restrict Search To** Limits the search to the CUI file you select from the drop-down list. By default, the main CUI file (*acad.cui* by default) is searched.

**Replace** Replaces a single instance of the search string. In the Customize User Interface dialog box, the location of the search string is displayed in the Command List pane, tree view, and Properties pane.

**Replace All** Replaces all instances where the search string is found. You cannot undo this operation.

**Find Next** Locates the next instance of the search string in the Name, Description, or Macros boxes in the Properties pane. If you search in the Command List pane, only commands in the list are located. If you search in the tree view in the Customizations In pane, the search starts in the tree view and continues to the commands in the Command List pane.

# CUIEXPORT

## Quick Reference

Exports customized settings to an enterprise or partial CUI file

Tools ➤ Customize ➤ Export CustomizationsAt the Command prompt, enter cuiexport.

**cuiexport**

Opens with the Transfer Tab (Customize User Interface Dialog Box) (page 276) opened by default. The main CUI file (*acad.cui*) is open in the left pane. You can drag items from one CUI file to the other. Click Apply to save the changes and view the updated CUI file.

# CUIIMPORT

## Quick Reference

Imports customized settings from an enterprise or partial CUI file to acad.cui

Tools ➤ Customize ➤ Import CustomizationsAt the Command prompt, enter cuiimport.

**cuiimport**

Opens with the Transfer Tab (Customize User Interface Dialog Box) (page 276) opened by default. You can drag items from one CUI file to the other. Click Apply to save the changes and view the updated CUI file.

# CUILOAD

## Quick Reference

Loads a CUI file

**cuiload**

Opens the Load/Unload Customizations dialog box (page 285), where you can locate and load a CUI file to customize or transfer user interface settings.

When *FILEDIA* is set to 0 (off), CUILOAD displays the following command prompt.

Enter name of customization file to load: *Enter a file name*

---

**NOTE** The Legacy Menu File (MNS) and Legacy Menu Template (MNU) files used in past releases have been replaced with just one file type, the XML-based CUI file.

---

# Load/Unload Customizations Dialog Box

## Quick Reference

**cuiload**

Controls the display of customization groups and interface elements.

**Loaded Customization Groups** Lists the currently loaded customization files.

**File Name** Specifies the file to load when you choose Load. You can either enter the file name or choose Browse to display the Select Customization File dialog box (a standard file selection dialog box (page 931)).

**Unload** Unloads the customization group selected in the Customization Groups list.

**Load** Loads the file specified under File Name.

**Browse** Displays the Select Menu File dialog box (a standard file selection dialog box (page 931)), in which you can select a menu file to load.

# CUIUNLOAD

## Quick Reference

Unloads a CUI file

**cuiunload**

Opens the Load/Unload Customizations dialog box (page 285), which has the same options as *CUILOAD*. The only difference between the two commands is in the command line prompts.

When *FILEDIA* is set to 0 (off), CUIUNLOAD displays the following command prompt.

Enter the name of a Customization Group to unload: *Enter a name*

# CUSTOMIZE

## Quick Reference

Customizes tool palettes and tool palette groups

Tools ➤ Customize ➤ Tool PalettesAt the Command prompt, enter customize.
Right-click any tool palette and choose Customize Palettes.
**customize**

The Customize dialog box (page 287) is displayed.

# Customize Dialog Box

## Quick Reference

Tools ➤ Customize ➤ Tool PalettesAt the Command prompt, enter customize.
Right-click any tool palette and choose Customize Palettes.
**customize**

Provides an interface for customizing tool palettes, palette groups, and block authoring palettes.

Creates, modifies, and organizes tool palettes and palette groups. Imports and exports palette files.

**Palettes** Lists all available tool palettes.

Click and drag a tool palette to move it up or down in the list. Right-click a palette in the list to rename, delete, or export the palette. (When you export a palette, it's saved to a file with an *.xtp* extension.) Right-click in the Palettes area to import a palette or to create a new, blank one.

The shortcut menus also provide options to

- Rename an existing palette
- Create a new palette
- Delete a tool palette
- Export a tool palette (as an XTP file)
- Import a tool palette

**Palette Groups** Displays the organization of your palettes in a tree view.

Click and drag a palette to move it into another palette group. Right-click a palette group, and then click Set Current on the shortcut menu to display the palette group.

The shortcut menus also provide options to

- Create a new palette group
- Delete an existing palette group
- Rename an existing palette group
- Remove a tool palette from a group

- Export a palette group (as an XPG file)
- Export all palette groups (as an XPG file)
- Import a palette group

**NOTE** You might need to collapse all palette groups to expose a blank area within the Palette Groups area. Then, right-click in the blank area to display shortcut menu options that are not specific to any existing palette groups.

**Current Palette Group** Displays the name of the palette group currently shown. Displays All Palettes when all available palettes are shown.

# CUTCLIP

## Quick Reference

Copies objects to the Clipboard and removes the objects from the drawing

Standard 
Edit ➤ CutAt the Command prompt, enter cutclip.
End any active commands, right-click in the drawing area, and choose Cut.
**cutclip**
Select objects:

CUTCLIP moves the selected objects to the Clipboard, removing them from the drawing. You can paste the contents of the Clipboard into a document or drawing as an embedded OLE object (see *PASTECLIP* and *PASTESPEC*). CUTCLIP does not create OLE link information.

# CYLINDER

## Quick Reference

Creates a three-sided 3D solid with a circular or elliptical base and top

Modeling 

Draw ➤ Modeling ➤ CylinderAt the Command prompt, enter cylinder.

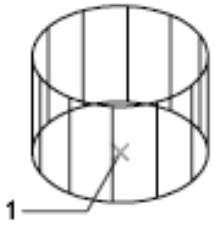
**cylinder**

3D Make panel, Cylinder

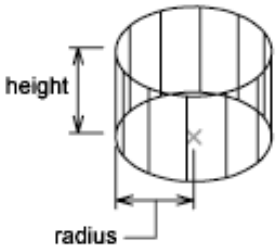
Specify center point of base or [3P (page 289)/2P (page 290)/Ttr (page 290)/Elliptical (page 291)]: *Specify a point (1) or enter an option*

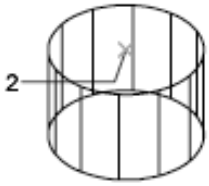
Specify base radius or [Diameter (page 292)] <default>: *Specify a base radius, or enter* **d** *to specify a diameter, or press* ENTER *to specify the default base radius value*

Specify height or [2Point (page 293)/Axis endpoint (page 293)] <default>: *Specify a height (2), enter an option, or press* ENTER *to specify the default height value*




Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.




### 3P (Three Points)

Defines the base circumference and base plane of the cylinder by specifying three points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*
Specify third point: *Specify a point*

Specify height or [2Point/Axis endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cylinder axis. This endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

### 2P (Two Points)

Defines the base diameter of the cylinder by specifying two points.

Specify first endpoint of diameter: *Specify a point*
Specify second endpoint of diameter: *Specify a point*
Specify height or [2Point/Axis endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cylinder axis. The axis endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

### TTR (Tangent, Tangent, Radius)

Defines the base of the cylinder with a specified radius tangent to two objects.

Specify point on object for first tangent: *Select a point on an object*
Specify point on object for second tangent: *Select a point on an object*
Specify base radius <default>: *Specify a base radius or press* ENTER *to specify the default base radius value*

Sometimes more than one base matches the specified criteria. The program draws the base of the specified radius whose tangent points are closest to the selected points.

Specify height or [2Point/Axis endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

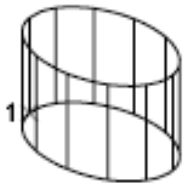
**Axis Endpoint** Specifies the endpoint location for the cylinder axis. The axis endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

### Elliptical

Specifies an elliptical base for the cylinder.

Specify endpoint of first axis or [Center]: *Specify a point (1)*
Specify other endpoint of first axis: *Specify a point*
Specify endpoint of second axis: *Specify a point*
Specify height or [2Point/Axis endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*



Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**Center** Creates the base of the cylinder by using a specified center point.

Specify center point: *Specify a point*

Specify distance to first axis <default>: *Specify a distance or press* ENTER *to specify the default distance value*

Specify endpoint of second axis: *Specify a point*

Specify height or [2Point/Axis endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cylinder axis. The axis endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

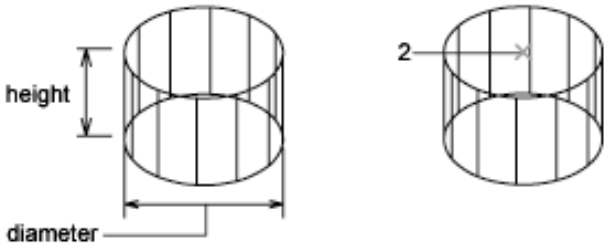
### Diameter

Specifies the diameter for the base of the cylinder.

Specify diameter <default>: *Specify a diameter or press* ENTER *to specify the default value*

Initially, the default diameter is not set to any value. During a drawing session, the default value for the diameter is always the previously entered diameter value for any solid primitive.

Specify height or [2Point/Axis endpoint] <default>: *Specify a height (2), enter an option, or press* ENTER *to specify the default height value*




Initially, the default height is not set to any value. During a drawing session, the default value for the height is always the previously entered height value for any solid primitive.

**2Point** Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*

Specify second point: *Specify a point*

**Axis Endpoint** Specifies the endpoint location for the cylinder axis. The axis endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

### 2Point

Specifies that the height of the cylinder is the distance between the two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

### Axis Endpoint

Specifies the endpoint location for the cylinder axis. The axis endpoint is the center point of the top face of the cylinder. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the cylinder.

Specify axis endpoint: *Specify a point*

# D Commands

# 5

**In this chapter**

- DIMBASELINE
- DIMBREAK
- DIMCENTER
- DIMCONTINUE
- DIMDIAMETER
- DIMDISASSOCIATE
- DIMEDIT
- DIMINSPECT
- DIMJOGGED
- DIMJOGLINE
- DIMLINEAR
- DIMORDINATE
- DIMOVERRIDE
- DIMRADIUS
- DIMREASSOCIATE
- DIMREGEN
- DIMSPACE
- DIMSTYLE
- DIMTEDIT
- DIST
- DISTANTLIGHT
- DIVIDE
- DONUT
- DRAGMODE
- DRAWINGRECOVERY
- DRAWINGRECOVERYHIDE
- DRAWORDER
- DSETTINGS
- DSVIEWER
- DVIEW

- DWFADJUST
- DWFATTACH
- DWFCLIP
- DWFLAYERS
- DWGPROPS
- DXBIN

# DASHBOARD

## Quick Reference

Opens the Dashboard window

Tools ➤ Palettes ➤ DashboardDoes not exist in the menus.
**dashboard**

The Dashboard window opens. The dashboard is organized into control panels that contain sets of related tools and controls. The number of control panels displayed on the dashboard depends on the current workspace, but can also be displayed or hidden using the dashboard shortcut menus.

The dashboard displays the control panels associated with the workspace you used last. To display the control panels associated with a specific workspace, click Tools ➤ Workspaces and select a workspace.

**NOTE** On the 3D Modeling dashboard, the 2D Draw control panel is initially hidden. Use any dashboard shortcut menu to display it when needed.

The large icons displayed at the left side of the dashboard are called control panel icons. Each control panel icon identifies the purpose of the control panel. On some control panels, if you click the icon, a slide-out panel opens that contains additional tools and controls. Only one slide-out panel can be open at a time.

You can click the dashboard Properties button or right-click the title bar, a control panel, a slide-out panel, or a control panel icon to display a shortcut menu. The options displayed on each shortcut menu are appropriate to the cursor location. All the options on these shortcut menus are listed and described below.

For information about customizing dashboard panels using the Customize User Interface dialog box, see Customize Dashboard Panels in the Customization Guide.

### Allow Docking

Controls whether the dashboard docks when you drag it over a docking area at the side of the drawing. Clear this option to undock the dashboard.

### Anchor Left

Docks and hides the dashboard on left side of the application window, taking precedence over other docked windows.

### Anchor Right

Docks and hides the dashboard on right side of the application window, taking precedence over other docked windows.

### Auto-hide

Controls display of the dashboard. When this option is selected, only the dashboard title bar is displayed when the cursor moves outside the dashboard. When this option is cleared, the full dashboard is always displayed.

### Close

Closes the Dashboard window.

### Control Panels

Lists the names of all control panels available for the dashboard. Click or clear each name to determine whether the associated control panel is displayed or hidden.

### Help

Displays this topic in the Help window.

### Hide

Hides the control panel on the dashboard.

### Move

Changes the cursor to a four-direction arrow.

### Show More Controls

Opens the slide out panel of the associated control panel.

#### Show Fewer Controls

Closes the slide out panel of the associated control panel.

#### Size

Changes the cursor to a four-direction arrow. Drag the edge opposite the title bar of the dashboard to change its horizontal size.

#### Tool Palette Group

Displays the list of available tool palette groups. Click a tool palette group to associate with the control panel. The associated tool palette group will display automatically when you click the control panel icon.

#### Transparency

Displays the Transparency dialog box (page 994).

# DASHBOARDCLOSE

## Quick Reference

Closes the Dashboard window

Tools ➤ Palettes ➤ DashboardDoes not exist in the menus.
**dashboardclose**

Closes the Dashboard window.

# DATAEXTRACTION

## Quick Reference

Exports object property, block attribute, and drawing information to a data extraction table or to an external file and specified a data link to an Excel spreadsheet

Modify II 

Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

The Data Extraction wizard is displayed.

If you enter **-dataextraction** at the command prompt, options are displayed at the command prompt. (page 325)

# Data Extraction Wizard

## Quick Reference

Modify II 
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

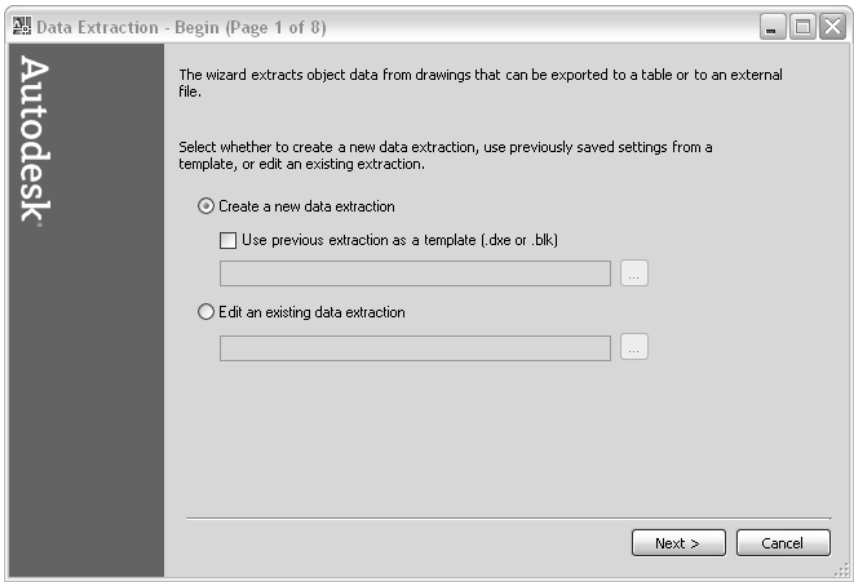
Provides step-by-step instructions for extracting information from objects, blocks, and attributes, including drawing information in the current drawing or a set of drawings. The information is used to create a data extraction table in the current drawing, or is saved to an external file, or both.

The Data Extraction wizard includes the following pages:

- Begin (page 301)
- Define Data Source (page 302)
- Select Objects (page 304)
- Select Properties (page 305)
- Refine Data (page 306)
- Choose Output (page 309)
- Table Style (page 310)
- Finish (page 311)

### Begin

Starts the data extraction process. Options include creating a new data extraction, using a template, or editing an existing data extraction.



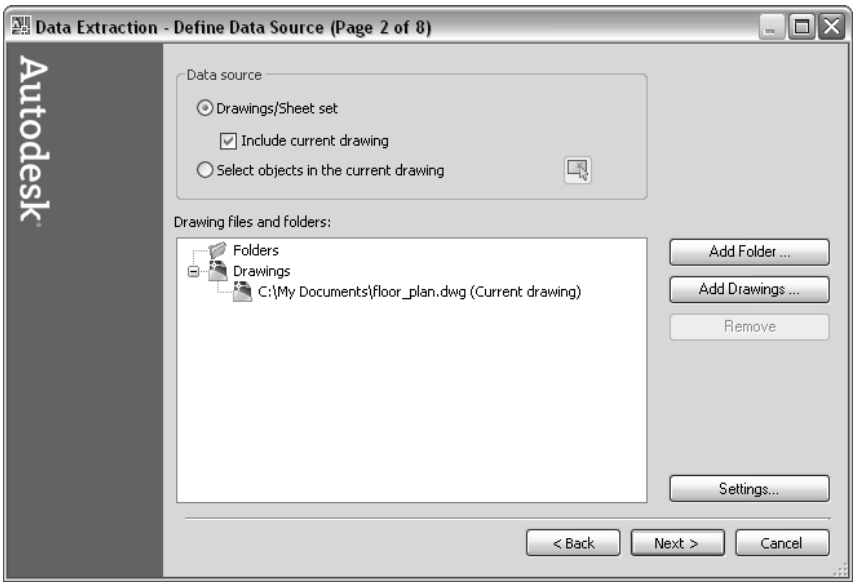

**Create a New Data Extraction** Creates a new data extraction and save it to a .DXE file. Also makes the Use a Previous Extraction as a Template button available so you can select a data extraction template (DXE) file or attribute extraction (BLK) file.

**Use a Previous Extraction as a Template** Uses settings previously saved in a data extraction (DXE) file or an attribute extraction template (BLK) file. As you move through the wizard, each page is already filled in with the settings in the template file. You can change these settings. Click the [...] button to select the file in a standard file selection dialog box (page 931).

**Edit an Existing Data Extraction** Allows you to modify an existing data extraction (DXE) file. Click the [...] button to select the data extraction file in a standard file selection dialog box (page 931).

### Define Data Source

Specifies the drawing files, including folders from which to extract data. Allows selection of objects in the current drawing from which to extract information.




## Data Source

**Drawings/Sheet Set** Makes the Add Folder and Add Drawings buttons available for specifying drawings and folders for the extraction. The drawings and folders for the extraction are listed in the Drawing Files view.

**Include Current Drawing** Includes the current drawing in the data extraction. The current drawing can be empty (not contain objects) if additional drawings are selected for extraction.

**Select Objects in the Current Drawing** Makes the Select Objects in the Current Drawing button available so you can select objects for data extraction.

**Select Objects button** Closes the wizard temporarily so that you select objects and blocks in the current drawing.

**Drawing Files and Folders** Lists the selected drawing files or folders. Checked folders are included in the extraction.

**Add Folder** Displays the Add Folder Options dialog box (page 312),where you can specify folders to include in the data extraction.

**Add Drawings** Displays the standard file selection dialog box (page 931), where you can specify drawings to include in the data extraction.

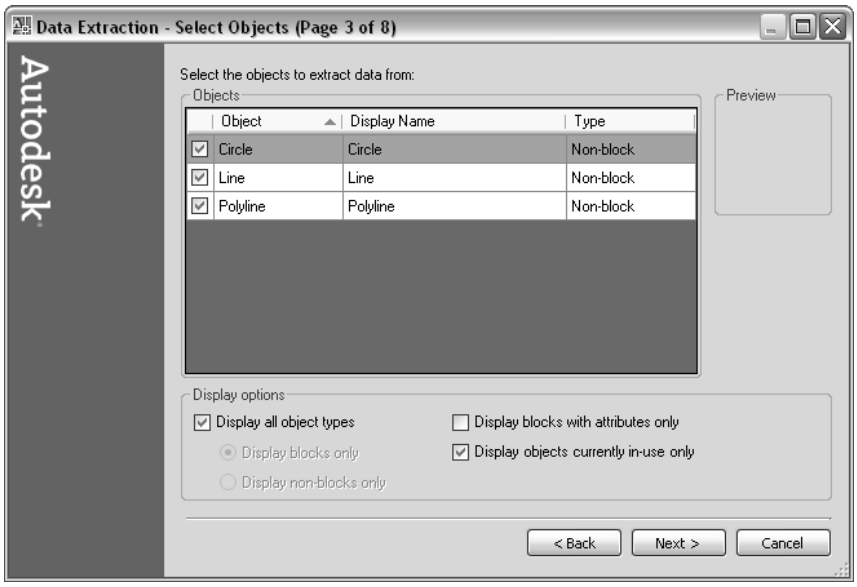
**Remove** Removes the checked drawings or folders listed in the Drawing Files and Folders list from the data extraction.

**Settings** Displays the Data Extraction - Additional Settings dialog box (page 313), where you can specify data extraction settings.

### Select Objects

Specifies the types of objects (blocks and non-blocks) and drawing information to be extracted.

Valid objects are checked by default. Objects that do not exist in the selected dawing are not checked. Click the column head to reverse the sort order. Columns can be resized. Property data from checked objects is displayed on the Select Properties (page 305) page.




**Object** Displays each object by its name. Blocks are listed by block name. Non-blocks are listed by their object name.

**Display Name** Provides a place to enter an optional alternative name for an object as it will appear in the extracted information. Select a display name, right-click in the list, and click Edit Display Name.

**Type** Displays whether the object is a block or non-block.

**Preview** Displays a preview image of the checked block in the Object list view.

### Display Options

**Display All Object Types** Displays a list of all object types (blocks and non-blocks) in the Object list view. This option is selected by default.

**Display Blocks Only** Displays only blocks in the Object list view.

**Display Non-Blocks Only** Displays only those objects that are not blocks in the Object list view.

**Display Blocks with Attributes Only** Displays only those blocks in the Object list view that contain attributes. Dynamic blocks are listed if they have special properties assigned to them (for example, actions and parameters).

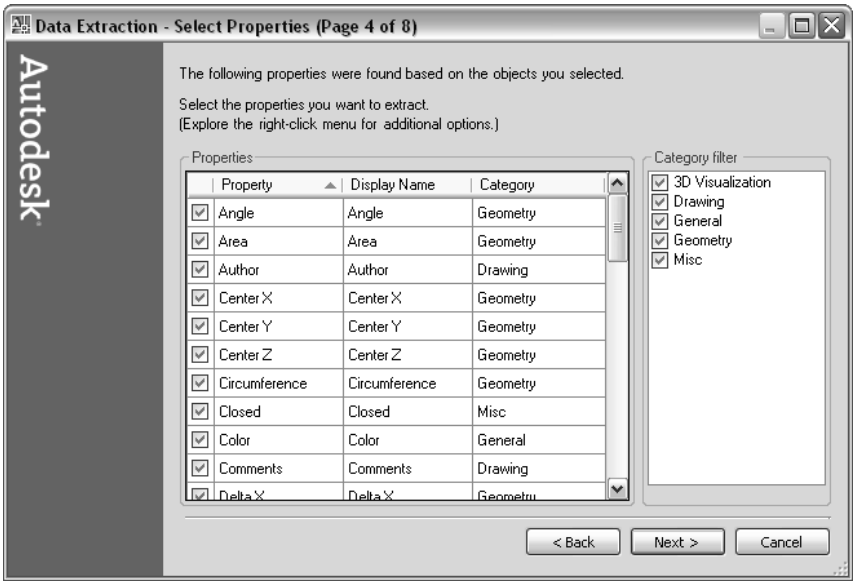
**Display Only Objects Used in Drawings** Displays objects in the Object list view that exist in the selected drawings.

### Select Properties

Controls the object, block, and drawing properties to extract.

Each row displays a property name, its display name, and category.

Right-click a column head and use options on the shortcut menu to check all or uncheck all items, invert the selection set, or edit the display name. Click the column head to reverse the sort order. Columns can be resized.



**Property** Displays object properties from objects selected on the  page. The property list is filtered according to the filter categories that are selected. Properties are the same as those listed in the Properties palette.

**Display Name** Provides a place to enter an optional alternative name for a property as it will appear in the extracted information. Select the property display name, right-click in the list, and click Edit Display Name.

**Category** Displays a category for each property. For example, *General* designates ordinary object properties, such as color or layer. *Attribute* designates user-defined attributes. *Dynamic* designates user-defined property data for dynamic blocks. Categories are the same as those listed in the Properties palette.

### Category Filter

**Property Category List** Displays a list of categories that are extracted from the property list. Unchecked categories filter the Properties list. Categories include 3D Visualization, Attributes, Drawing, Dynamic Blocks, General, Geometry, Misc, Pattern, Table, and Text.

### Property Grid Shortcut Menu

**Check All** Checks all properties in the property grid.

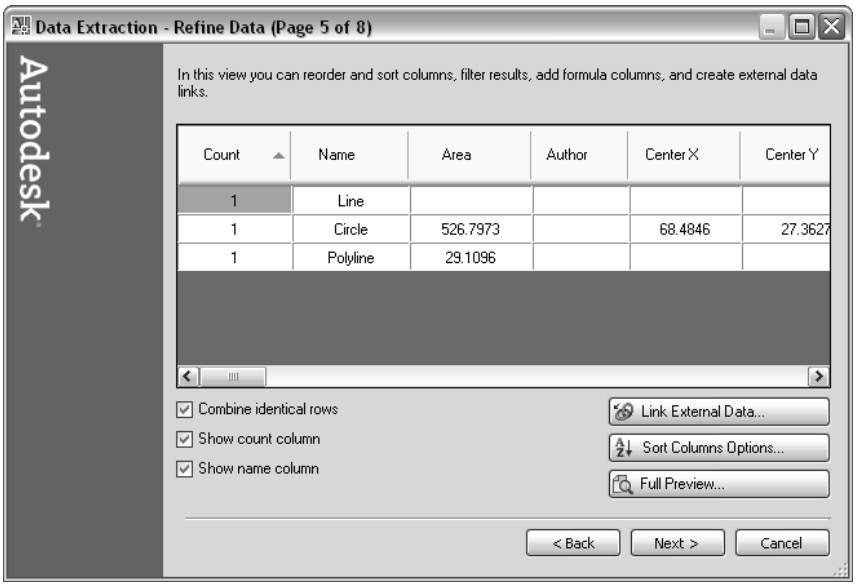
**Uncheck All** Unchecks all properties in the property grid.

**Invert Selection** Inverts the current checked items in the property grid.

**Edit Display Name** Allows in-place editing of the property display name for the selected row.

#### Refine Data

Modifies the structure of the data extraction table. You can reorder and sort columns, filter results, add formula columns and footer rows, and create a link to data in a Microsoft Excel spreadsheet.




### Data Grid View

**Columns** Displays properties in a columnar format as they were specified in the Select Properties page. Right-click any column header to display a shortcut menu of options. The Count column and Name column display by default. Icons display in the column header for inserted formula columns and columns extracted from a Microsoft Excel spreadsheet.

**Combine Identical Rows** Groups identical records by row in the table. Updates the Count column with the sum of all aggregated objects.

**Show Count Column** Displays the Count column in the grid.

**Show Name Column** Displays the Name column in the grid.

**Link External Data** Displays the Link External Data dialog box (page 315), where you can create a link between the extracted drawing data and data in an Excel spreadsheet.

**Sort Columns Options** Displays the Sort Columns dialog box (page 317), where you can sort data across multiple columns.

**Full Preview** Displays a full preview of the final output, including linked external data, in the text window. The preview is for viewing only.

### Column Shortcut Menu

**Sort Descending** Sorts column data in a descending order.

**Sort Ascending** Sorts column data in an ascending order.

**Sort Column Options** Displays the Sort Columns dialog box (page 317), where you can sort data across multiple columns.

**Rename Column** Allows in-place editing of the selected column name.

**Hide Column** Hides the selected column.

**Show Hidden Columns** Displays the hidden column. The flyout option includes Display All Hidden Columns.

**Set Column Data Format** Displays the Set Cell Format dialog box, (page 319) where you can set a data type for cells in the selected column.

**Insert Formula Column** Displays the Insert Formula Column dialog box (page 318), where you can specify the formula that is inserted into the table. Inserts the formula column to the right of the selected column. An existing formula column cannot be used as a value for another formula column.

**Edit Formula Column** Displays the Edit Formula Column dialog box (page 323). This option is only available when a formula column is selected.

**Remove Formula Column** Removes the selected formula column. This option is only available when a formula column is selected.

**Combine Record Mode** Displays numeric data in the selected column as separate values or collapses identical property rows into one row and displays the sum of all the numeric data in the selected column. This option is available when the Combine Identical Rows is checked and the selected column contains numerical data.

**Show Count Column** Displays a Count column that lists the quantity of each property.

**Show Name Column** Displays a Name column that displays the name of each property.

**Insert Totals Footer** Displays a flyout menu with options for Sum, Max, Min, and Average. Creates a footer row for the selected column that is placed below all the data rows and displays values based on the selected arithmetic function. This option is available only for columns that have a numeric data type.

**Sum** Displays a sum of all the values in the selected column in a footer row.

**Max** Displays the maximum value in the selected column in a footer row.

**Min** Displays the minimum value in the selected column in a footer row.

**Average** Displays the average value in the selected column in a footer row.

**Remove Totals Footer** Removes the Totals footer. This option is available when a footer row exists.

**Filter Options** Displays the Filter Column dialog box (page 324), where you can specify filter conditions for the selected column.

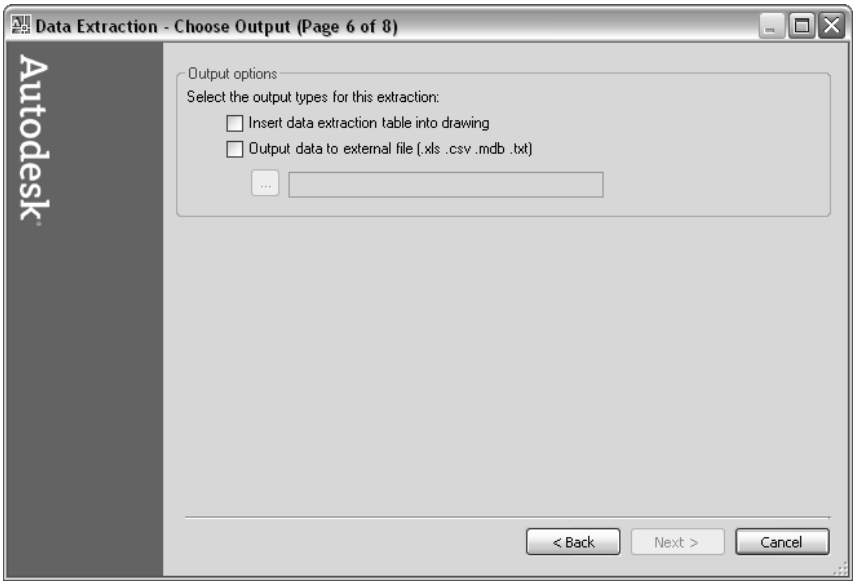
**Reset Filter** Restores the default filter for the selected column.

**Reset All Filters** Restores default filters for all columns that have filters.

**Copy to Clipboard** Copies all the data cells, including column names, to the Clipboard.

### Choose Output

Specifies the type of output to which the data is extracted.




### Output Options

**Insert Data Extraction Table into Drawing** Creates a table that is populated with extracted data. You are prompted to insert the table into the current drawing when you click Finish on the Finish (page 311) page.

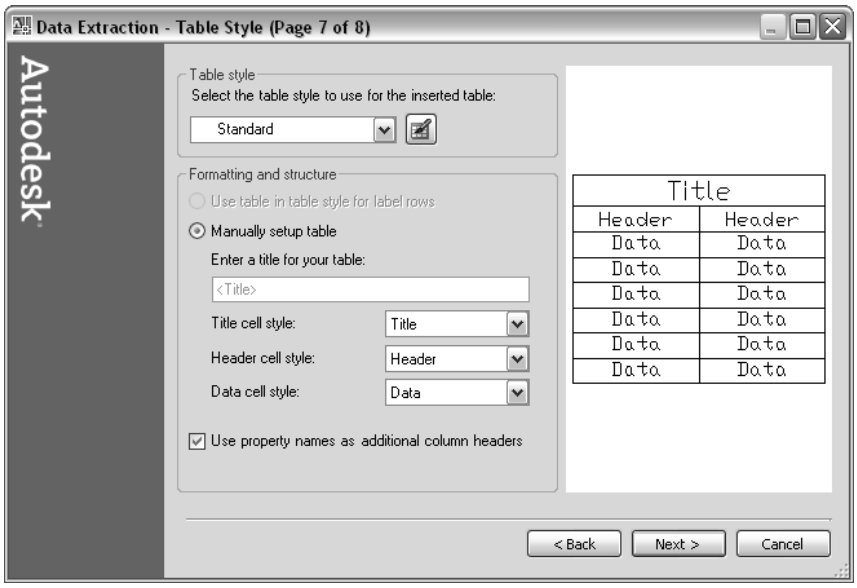
**Output Data to External File** Creates a data extraction file. Click the [...] button to select the file format in a standard file selection dialog box (page 931). Available file formats are Microsoft Excel (XLS), comma-separated file

format (CSV), Microsoft Access (MDB), and tab-separated file format (TXT). The external file is created when you click Finish on the Finish page.

The maximum number of columns that can be exported to an XLS and MDB file is 255.

### Table Style

Controls the appearance of the data extraction table. This page is displayed only if AutoCAD Table is selected on the Choose Output (page 309) page.




## Table Style

**Select the Table Style to Use for the Inserted Table** Specifies the table style. Click the Table Style button to display the Talbe Style dialog box (page 1402), or select a table style from the drop-down list that is defined in the drawing.

## Formatting and Structure

**Use Table in Table Style for Label Rows** Creates the data extraction table with a set of top rows that contain label cells and a bottom set of label rows that contain header and footer cells. Extracted data is inserted between the top and bottom label rows. This option is only available when the selected table style contains a template table.

**Manually Setup Table** Provides for manualy entering a title and specification of the title, header, and data cells style.

**Enter a Title for Your Table** Specifies a title for the table. This row is not overwritten when the table is updated. The default table style, STANDARD, includes a title row. If the selected table style does not include a title row, this option is not available.

**Title Cell Style** Specifies the style for the title cell. Click the drop-down list to select a title cell style defined in the selected table style.

**Header Cell Style** Specifies the style for the header row. Click the drop-down list to select a cell style defined in the selected table style.

**Data Cell Style** Specifies the style for data cells. Click the drop-down list to select a cell style defined in the selected table style.

**Use Property Names as Additional Column Headers** Includes column headers and uses the Display Name as the header row.

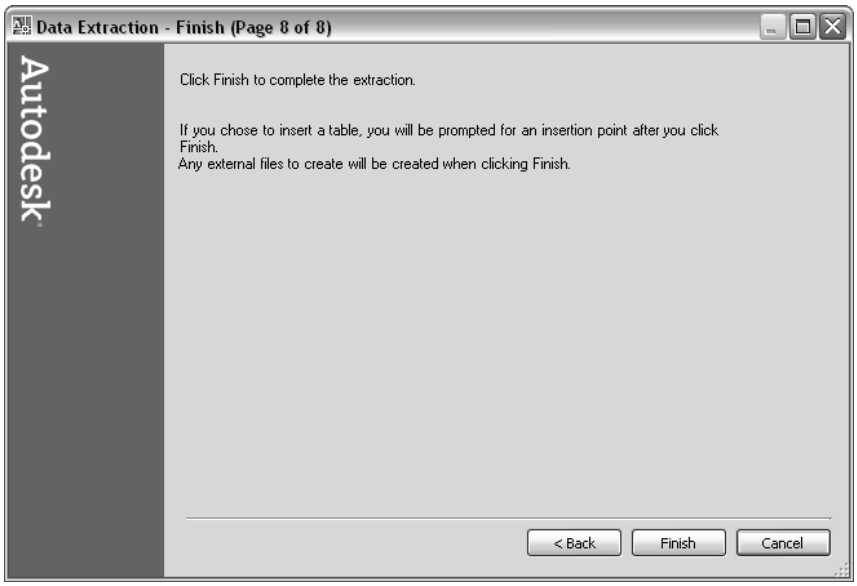
**Display Preview** Displays a preview of the table layout. If the table style does not include a title row or header row, none is displayed.

### Finish

Completes the process of extracting object property data that was specified in the wizard and creates the output type that was specified on the Choose Output (page 309) page. If data linking and column matching to an Excel spreadsheet was defined in the Link External Data dialog box (page 315), the selected data in the spreadsheet is also extracted.

If the Insert Data Extraction Table into Drawing option was selected on the Choose Output page, you are prompted to insert the table into the drawing when you click Finish.

If the Output Data to External File option was selected, the extracted data is saved to the specified file type.

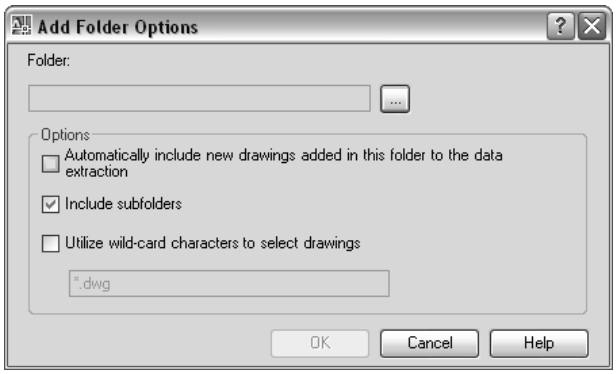



# Add Folder Options Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Specifies the folders to be used for data extraction. Drawings in the selected folders are monitored for data changes.




**Folder** Displays the path to the specified folder.

**Folder Button** Click the [...] button to select the folder in a standard file selection dialog box (page 931).

### Options

**Automatically Include New Drawings Added in this Folder to the Data Extraction** Includes new drawings to the data extraction when they are added to the folder. The New Drawings Found dialog box (page 314) displays when new drawings are added. When this option is selected, the specified folders are "live." When this option is not selected, the folders are "static."

**Include Subfolders** Includes drawings in subfolders of selected folders in the data extraction process.

**Utilize a Wild-card Character to Select Drawings** Activates a text entry field where you can enter search criteria using wild-card characters to select specific drawings.

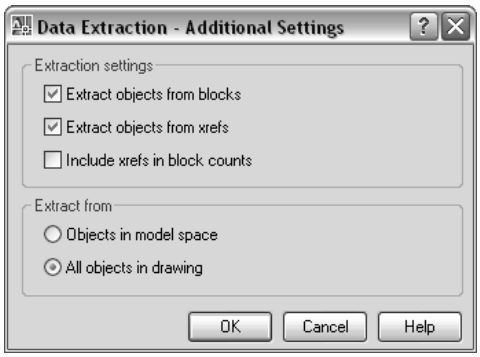
**Wild-Card Field** Enter wild-card characters. Valid characters are * ? .

## Data Extraction - Additional Settings Dialog Box

### Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Provides options for extracting objects in nested and xref blocks, options for counting blocks, and whether all objects are extracted or only those in model space.




### Extraction Settings

**Extract Objects from Blocks** Includes objects nested in blocks.

**Extract Objects from Xrefs** Includes objects and blocks in externally referenced (xref) files.

**Include Xrefs in Block Counts** Includes xrefs when counting blocks.

### Extract From

**Objects in Model Space** Includes only those objects in model space and ignores objects in paper space.

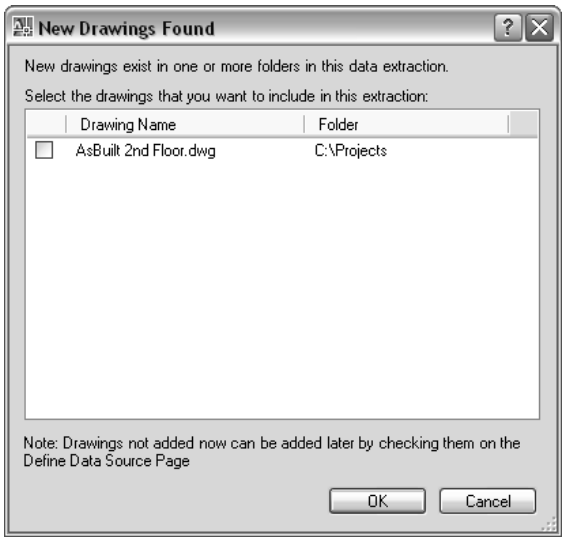
**All Objects in Drawing** Includes all objects in model space and paper space in the drawing, including drawing information. On by default.

## New Drawings Found Dialog Box

### Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Displays a list of new drawings that were added to the folder that was selected for the data extraction after the extraction was performed.




**Drawing Name** Displays the name of the drawing that was added to the data extraction folder.

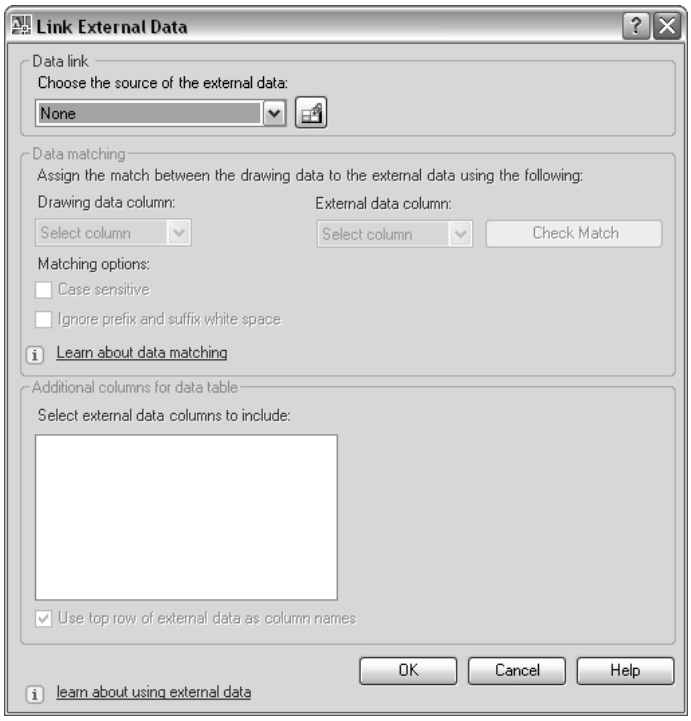
**Folder** Displays the path and folder name for each new drawing.

# Link External Data Dialog Box

## Quick Reference

Modify II 
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Matches the extracted drawing data to a Microsoft Excel spreadsheet through a data link and data matching.




### Data Link

**Choose the Source of the External Data** Displays a list of established links to an Excel spreadsheet.

**Data Link Manager Button** Displays the Data Link Manager (page 326), where you can specify a link to a Microsoft Excel spreadsheet.

### Data Matching

**Drawing Data Column** Displays a list of extracted property columns.

**External Data Column** Displays a list of column names from the linked data in the Excel spreadsheet.

**Check Match** Compares the data in the specified drawing data key column and the external data key column. If the data is unique across all rows in the external data key columns, and there is at least one data match between the drawing data and in data in the spreadsheet, a message displays the check key is successful. If the check key is unsuccessful, a warning message is displayed.

**Matching Options** Provides two options for the Check Match function: Case Sensitive and Ignore Prefix and Suffix White Space.

**Case Sensitive** When Case Sensitive is selected, the case of the data is checked.

**Ignore Prefix and Suffix White Space** Removes blank spaces in front of and after each data source before the check matching process proceeds. When this option is clear, white spaces are evaluated.

**Learn About Data Matching** Displays Understand Data Linking and Matching topic in the User's Guide.

#### Additional Columns for Data Table

**Select External Data Columns to Include** Displays the column names in the order in which they appear in the spreadsheet. Selected columns are linked to the drawing data through data matching.

**Use Top Row of External Data as Column Names** Uses the top row of data in the data link as the column name for the linked data. Otherwise, column names are labeled "Column 1, Column 2" and so on.

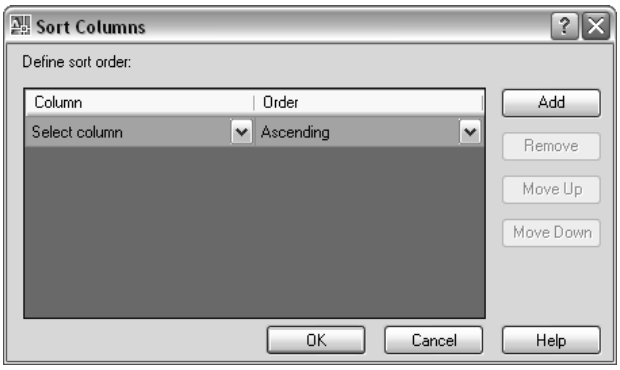
**Learn About Using External Data** Displays the New Features Workshop.

# Sort Columns Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Specifies a sort order for columns.




**Define Sort Order** Specifies the sort order of columns. The Column section contains column names from the Refine Data (page 306) page.

**Add** Adds a new item of sorting criteria to the Column list.

**Remove** Removes the currently selected sort criteria.

**Move Up** Moves the selected item up in the list view.

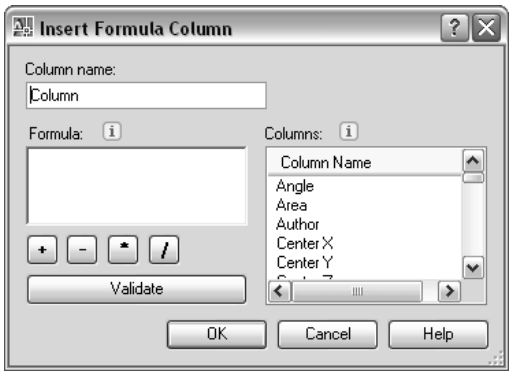
**Move Down** Moves the selected item down in the list view.

# Insert Formula Column Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Specifies the formula for the column that is inserted into the extracted data.




**Column Name** Specifies a name for the column that can be edited. Duplicate column names cannot be used.

**Formula** Displays the selected formula or functions that are available for QUICKCALC (page 1155) and CAL (page 211).

**Addition (+) Button** Adds the numerical data from one column to another.

**Minus (-) Button** Subtracts the numerical data from one column to another.

**Multiply (*) Button** Multiplies the numerical data from one column to another.

**Division (/) Button** Divides the numerical data from one column to another.

**Validate** Checks the validity of the specified equation. A formula column can only be inserted when the formula is validated.

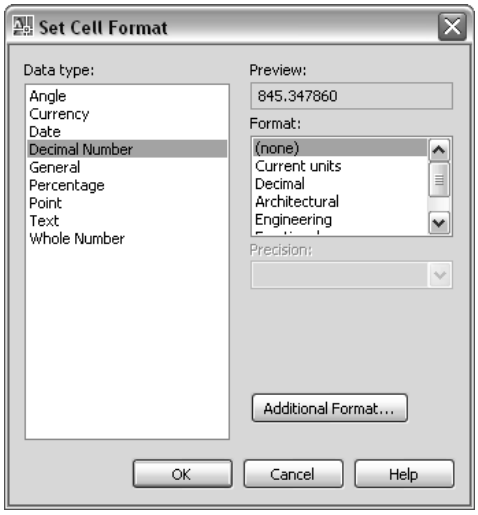
**Columns** Displays the column names from the extracted drawing data and external data (if data linking was established). Column names can be double-cked to be added to the Formula field or dragged from the Column list to the Formula field. Existing formula columns are not listed and cannot be used as values for creating additional formula columns.

# Set Cell Format Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**dataextraction**

Defines the formatting for cells in the table.




### Data Type

Displays a list of data types (Angle, Date, Decimal Number, and so on) that you can format for table rows.

**Preview** Displays a preview of the option you selected in the Format list.

**Format** Depending on the data type you select, displays a list of relevant format types. For example, if you select Angle as the data type, Format includes options such as Decimal Degrees, Grads, Radians, and so on.

**Precision** For Angle, Decimal Number, and Points data types only, sets the precision for applicable formats. For example, if you select Angle as the data type and Radians as the format type, Precision includes options such as Current Precision, 0.0r, 0.00r, 0.000r, and so on.

**List Separator** For a Point data type only, displays a list of options (comma, semicolon, or colon) that you can use to separate list items.

**Symbol** For Currency data types only, displays a list of currency symbols that you can use.

**Append Symbol** For Currency data types, places the currency symbol after the number. For Percentage data types, the percent symbol is placed after the number.

**Negative Numbers** For Currency data types only, lists options for displaying negative numbers.

**X, Y, and Z Coordinates** For Point data types only, filters X, Y, or Z coordinates.

**Additional Format** For Angle, Decimal Number, Point, and Whole Number data types only, opens the Additional Format dialog box (page 321), where you set additional formatting options for table cells.

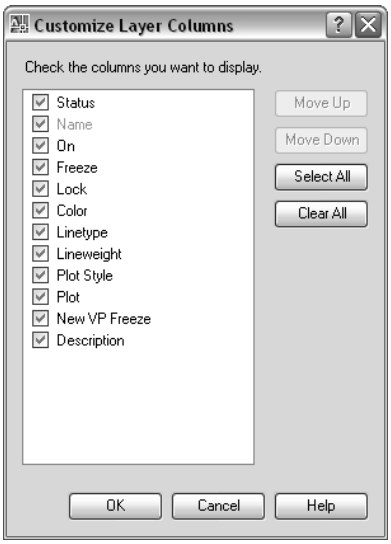
**Examples** For the Date data types only, displays a list of date display options in the Format field. Click a date in the Format field to see an example.

# Additional Format Dialog Box

## Quick Reference

Format ➤ Data Extraction

Provides additional formatting options for fields and table cells.




### Current Value

Displays the value in base drawing units.

### Preview

Displays updates to the format as you change the conversion factor and other settings.

### Conversion Factor

Specifies the conversion factor to use on the the current value. The default is 1 for no conversion.

### Additional Text

Specifies a prefix or a suffix for the value.

### Number Separators

Specifies a decimal separator and the formatting for numbers over 1000.

**Decimal** Specifies the separator for decimal values. Select a period, a comma, or a space.

**Thousands** Inserts a comma to group thousands in a field value.

### Zero Suppression

Controls the suppression of leading and trailing zeros, and of feet and inches that have a value of zero.

**Leading** Suppresses leading zeros in all decimal values. For example, 0.5000 becomes .5000.

**Trailing** Suppresses trailing zeros in all decimal values. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

**0 Feet** Suppresses the feet portion of a feet-and-inches value when the distance is less than one foot. For example, 0'-6 1/2" becomes 6 1/2".

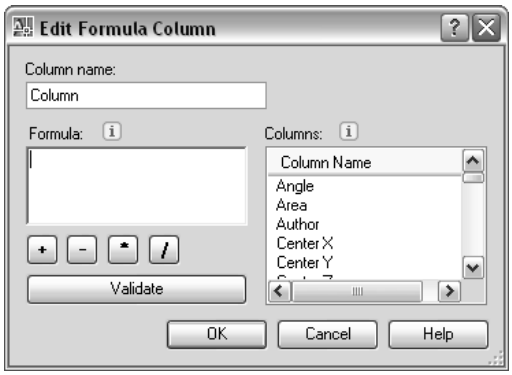
**0 Inches** Suppresses the inches portion of a feet-and-inches value when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

# Edit Formula Column Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
dataextraction

Displays the equation for the selected formula column.




**Column Name** Specifies a name for the column that can be edited.

**Formula** Displays the existing formula or function..

**Plus (+)** Adds the numerical data from one column to another.

**Minus (-)** Subtracts the numerical data from one column to another.

**Multiply (*)** Multiplies the numerical data from one column to another.

**Division (/)** Divides the numerical data from one column to another.

**Validate** Checks the validity of the specified equation. A formula column can only be inserted when the formula is validated.

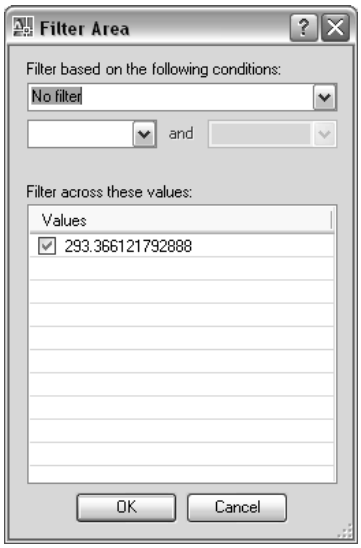
**Columns** Displays the column names from the extracted drawing data and external data (if data linking was established). Column names can be double-clicked to be added to the Formula field or dragged from the Column list to the Formula field.

# Filter Column Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
dataextraction

Filters column data by specified filters.




**Filter Based on the Following Conditions** Displays conditions based on the type of data in the selected column.

**First Column** Specifies the first condition.

**Second Column** Available when the specified filter uses two conditions.

**Filter Across These Values** Displays a list of values depending on the type of data being filtered. Filters for numeric data include Greater than >, Equal To, Between, and so on. Filters for textual data include Equal To, Not Equal To, Contains, and Begins With.

# Data Extraction - Out of Date Table Dialog Box

## Quick Reference

Modify II
Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
dataextraction

Offers options to update the data extraction table, update all tables, or skip the update.


**Update** Updates the data extraction table in the current drawing.

**Update All** Updates all the data extraction tables.

**Skip Update** Does not update the table.

# -DATAEXTRACTION

## Quick Reference

If you enter **-dataextraction** at the command prompt, the following DATAEXTRACTION command prompts are displayed.

Extracts data as specified in an existing attribute extraction template (BLK) file created with the Attribute Extraction wizard in AutoCAD 2006 or data extraction (DXE) file created in AutoCAD 2007.

Enter the template file path for the extraction: type: *Specify the path and file name for the attribute extraction template (BLK) or data extractino (DXE) file that describes how to extract the information*

Subsequent prompts depend on the information set up in the template file. If the template specifies extracting data to an external file, the following prompts are displayed:

Enter the output filetype [Csv/Txt/Xls/Mdb] <Csv>: *Enter* **c** *for comma-separated (CSV),* **t** *for tab-separated (TXT),* **x** *for Microsoft Excel (XLS), or* **m** *for Microsoft Access (MDB)*
Enter output filepath: *Specify the names of the path and file where the data will be extracted*

**NOTE** The maximum number of columns that can be exported to an XLS and MDB file is 255.

If the template specifies a table, the following prompt is displayed:

Specify insertion point:

# DATALINK

## Quick Reference

The Data Link Manager is displayed.

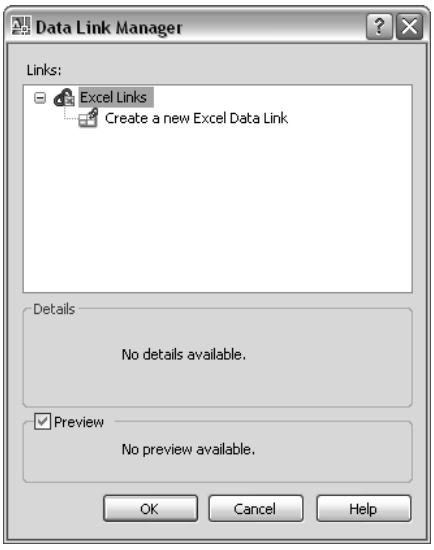
Tools ➤ Data Links ➤ Data Link ManagerDoes not exist in the menus.
Tables panel, Data Link Manager
**datalink**

The Data Link Manager (page 326) is displayed.

# Data Link Manager

## Quick Reference

Creates, edits, and manages data links.




Tools ➤ Data Links ➤ Data Link ManagerDoes not exist in the menus.
**datalink**

### Data Link Tree View

Displays links contained within the drawing. Also gives options for creating new data links.

**Excel Links** Lists the Microsoft Excel data links within the drawing. If the icon displays a linked chain, then the data link is valid. If the icon displays a broken chain, then the data link is broken.

**Create a New Excel Data Link** Launches a dialog box where you enter the name for a new data link. Once a name has been created, the New Excel Data Link dialog box (page 328) is displayed.

### Details

Lists information for the data link selected in the tree view above.

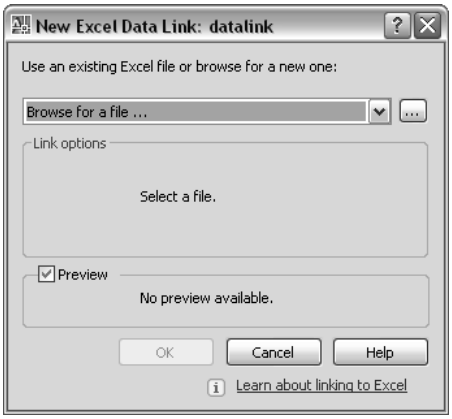
### Preview

Displays a preview of the linked data as it would appear in the drawing table. When a data link is not currently selected then no preview is displayed.

# New and Modify Excel Link Dialog Box

## Quick Reference

Links data from a spreadsheet created in Microsoft Excel to a table within your drawing.




**datalink**

### Data Link Drop-Down Menu

Allows you to choose an established Microsoft XLS, XLSX, or CSV file to link to your drawing. At the bottom of this drop-down list, you can select a new XLS, XLSX, or CSV file from which to create a data link.

Click the [...] button to browse for another Microsoft Excel file on your computer.

### Link Options

Specifies the data in your Excel file to link to your drawing.

**Select the Excel Sheet to Link to** Displays the names of all sheets within the specified XLS, XLSX, or CSV file. The link options specified below are applied to the sheet you choose here.

**Link whole sheet** Links the entire specified sheet within your Excel file to a table in your drawing.

**Link to a named range** Links a named range of cells already contained within your Excel file to a table in your drawing.

Clicking the arrow displays the available named ranges found in the linked spreadsheet.

**Link to range** Specifies a range of cells in your Excel file to link to a table in your drawing.

In the text box, enter the range of cells you want linked to your drawing. Valid ranges include

- Rectangular regions (for example, A1:D10)
- Entire columns (for example, A:A)
- Sets of columns (for example, A:D)

Click the button to the right of the text box to preview the linked range.

### Preview Window

Displays a preview of your table using the options you have applied.

### Cell Contents

Options in this box will determine how data is imported into your drawing from your external source.

**Convert Data Types to Text** Imports Microsoft Excel data as text with data calculated from formulas in Excel (supported data formats not attached).

**Retain Formulas** Imports data with formulas and supported data formats attached.

**Allow Writing to Source File** Specifies that the DATALINKUPDATE (page 330) command can be used to upload any changes made to linked data in your drawing to the original external spreadsheet.

### Cell Formatting

**Use Excel Formatting** Imports any formatting specified in the original XLS, XLSX, or CSV file will be brought into your drawing.

**Keep Table Updated to Excel Formatting** If the option above is selected, updates any changed formatting when the DATALINKUPDATE (page 330) command is used.

**Start With Excel Formatting, Do Not Update** Imports the formatting specified in the original XLS, XLSX, or CSV file into your drawing, but any changes made to the formatting are not included when the DATALINKUPDATE (page 330) command is used.

# DATALINKUPDATE

## Quick Reference

Updates data to or from an established external data link.

Tools ➤ Data Links ➤ Update Data LinksDoes not exist in the menus.
Tables panel, Update Data Links
**datalinkupdate**

Updates data in a drawing that is referenced by changed data in an external data link. Also updates any data in an external data link that has been changed within a linked table in your drawing.

Select an option [Update data link/Write data link] <Update data link>:

### Update Data Link

Updates data in a drawing that is referenced by changed data in an established data link.

Select objects or [Data link/All]:
**Select Objects** Updates tables containing data links with data that has been changed in the external source file.

**Data Link** Specifies the name of a data link to update the link with data that has been changed in the external source file.

Enter data link name or [?]:

Entering [?] lists the data links in the current drawing.

**All** Updates all data links in all tables in the drawing with data that has been changed in the external source file.

### Write Data Link

Updates any data in an external data link that has been changed within a linked table in your drawing.

Select objects:

**Select Objects** Uploads data that has been changed from the original linked content to the source file.

# DBCONNECT

## Quick Reference

Provides an interface to external database tables

Tools ➤ Palettes ➤ dbConnect
**dbconnect**

The dbConnect Manager (page 332) is displayed and the dbConnect menu is added to the menu bar.

The four primary interfaces (the dbConnect Manager, the Data View window, the Query Editor, and the Link Select dialog box) are described first, followed by descriptions of each additional dialog box presented in alphabetical order. The Column Values dialog box, a sub-dialog box of the Query Editor, is described in the Query Editor section. The section describing each dialog box contains a list of methods that you can use to access it.

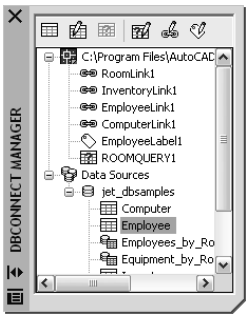
- dbConnect Manager (page 332)
- Data View window (page 337)
- Query Editor (page 346)
- Column Values dialog box (page 353)
- Link Select dialog box (page 369)
- Configure a Data Source dialog box (page 354)
- Data View and Query Options dialog box (page 354)
- Export Links dialog box (page 356)
- Export Query Set dialog box (page 357)
- Export Template Set dialog box (page 358)
- Find dialog box (page 358)
- Format dialog box (page 359)
- Import Query Set dialog box (page 359)

- Import Template Set dialog box (page 360)
- Label Template dialog box (page 360)
- Label Template Properties dialog box (page 365)
- Link Conversion dialog box (page 366)
- Link Manager (page 369)
- Link Template dialog box (page 372)
- Link Template Properties dialog box (page 374)
- New Label Template dialog box (page 375)
- New Link Template dialog box (page 375)
- New Query dialog box (page 376)
- Replace dialog box (page 377)
- Select a Database Object dialog box (page 378)
- Select Data Object dialog box (page 378)
- Sort dialog box (page 379)
- Synchronize dialog box (page 379)

# dbConnect Manager

## Quick Reference

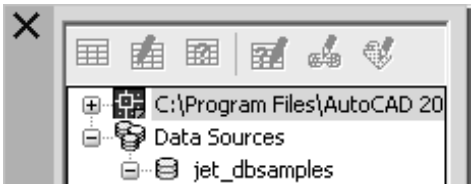
Provides the primary interface for the connectivity feature. You can view and edit database tables, execute structured query language (SQL) queries, and link table records to graphical objects. By default, the dbConnect Manager opens as a docked window on the left side of the drawing area.




When you open the dbConnect Manager, a small red X is displayed in the lower right corner of each database and data object. The X indicates that these objects are currently unconnected. To connect to a database or data object, double-click it in the dbConnect Manager.

### dbConnect Manager Buttons

Display and manipulate database objects.




**View Table** Opens an external database table in Read-only mode. This button is not available unless a single table, link template, or label template is selected in the tree view.

**Edit Table** Opens an external database table in Edit mode. This button is not available unless a single table, link template, or label template is selected in the tree view.

**Execute Query** Executes a query. This button is not available unless a query is selected in the tree view.

**New Query** Displays the New Query dialog box (page 376). This button is not available unless a single table, link template, or query is selected. If a query is selected, you can use this button to display the Query Editor (page 346), in which you can edit the query.

**New Link Template** Displays the New Link Template dialog box (page 375). This button is not available unless a single table or link template is selected. If a link template is selected, you can use this button to display the Link

Template dialog box (page 372), in which you can edit the link template. Not available for link templates with links already defined in a drawing.

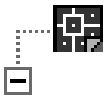
**New Label Template** Displays the New Label Template dialog box (page 375). This button is not available unless a single table, link template, or label template is selected. If a label template is selected, you can use this button to display the Label Template dialog box (page 360), in which you can edit the label template.

### dbConnect Manager Tree View

Contains nodes for each drawing that's currently open and a Data Sources node that contains all available data sources configured on your system.

### Drawing Nodes Shortcut Menu

Provides options available for an open drawing. Right-click the node of an open drawing.



**Export Template Set** Opens the Export Template Set dialog box (page 358), in which you can save all link and label templates stored in the drawing to an external file.

**Import Template Set** Opens the Import Template Set dialog box (page 360), in which you can import a set of link and label templates that are stored in an external file.

**Export Query Set** Opens the Export Query Set dialog box (page 357), in which you can save all queries stored in the drawing to an external file. The exported query set is saved with the file extension *.dbq*.

**Import Query Set** Opens the Import Query Set dialog box (page 359), in which you can import a set of queries stored in an external file with the file extension *.dbq*.

**Show Labels** Turns on visibility of all labels in the selected drawing.

**Hide Labels** Turns off visibility of all labels in the selected drawing.

**Reload Labels** Refreshes the field values of the labels in the selected drawing, updating them to reflect any changes made in the source database table.

### Database Objects Shortcut Menu

Provides options available for database objects (such as link templates, label templates, and queries) attached to the drawing nodes. Different database objects have different subsets of menu options available to them. For example, the shortcut menu options that are displayed when you right-click a query differ from the options that are available when you right-click a link template. The following shortcut menu options are available for various database objects.

**View Table** Opens an external database table in Read-only mode. Available only for link templates.

**Edit Table** Opens an external database table in Edit mode. Available only for link templates.

**Edit** Opens a dialog box in which you can edit the properties of the selected database object. Available for link templates, label templates, and queries. Not available for link templates with links already defined in the drawing.

**Delete** Deletes the selected database object. Available for link templates, label templates, and queries.

**Duplicate** Makes a copy of the selected database object and inserts it in the current drawing. Available for link templates, label templates, and queries.

**Rename** Opens a dialog box in which you can rename the selected database object. Available for link templates, label templates, and queries. Not available for link templates with links already defined in the drawing.

**New Query** Opens the New Query dialog box (page 376). Available only for link templates.

**Link Select** Opens the Link Select dialog box (page 369). Available only for link templates.

**New Label Template** Opens the New Label Template dialog box (page 375). Available only for link templates.

**Show Labels** Displays all labels that are associated with the selected database object. Available for link templates and label templates.

**Hide Labels** Hides all labels that are associated with the selected database object. Available for link templates and label templates.

**Delete Links** Deletes all links in the current drawing that use the selected link template. Available only for link templates.

**Delete Labels** Deletes all labels in the current drawing that use the selected label template. Available only for label templates.

**Synchronize** Verifies that all links in the current drawing based on the selected link template contain valid values. Any detected problems are reported in the Synchronize dialog box (page 379). Available only for link templates.

**Reload** Refreshes the field values of all labels associated with the selected label template in the current drawing, updating them to reflect any changes made in the source database table. Available only for label templates.

**Execute** Executes the selected query. Available only for queries.

**Delete Links** Deletes all links in the current drawing that use the selected link template. Available only for link templates.

### Data Sources Node Shortcut Menu

Provides options available for data sources. Right-click the data sources node.

**Configure Data Source** Opens the Configure a Data Source dialog box (page 354) where you can configure a new data source or edit an existing one.

### Data Objects Shortcut Menu

Provides options available for data objects. Right-click a data object (such as a catalog or table) in the data sources node.

**Connect** Establishes a connection to the selected data object. Available for data sources, catalogs, and schemas.

**Disconnect** Closes the connection to the selected data object. Available for data sources, catalogs, and schemas.

**Synchronize** Opens the Synchronize dialog box (page 379). Available only for connected data sources.

**Configure** Opens the Configure a Data Source dialog box (page 354) where you can configure a new data source or edit an existing one. Available only for disconnected data sources.

**View Table** Opens the selected database table in Read-only mode. Available only for tables.

**Edit Table** Opens the selected database table in Edit mode. Available only for tables.

**New Link Template** Opens the New Link Template dialog box (page 375). Available only for database tables.

**New Label Template** Opens the New Label Template dialog box (page 375). Available only for database tables.

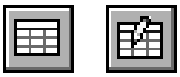
**New Query** Opens the New Query dialog box (page 376). Available only for database tables.

# Data View Window

## Quick Reference

Provides the primary interface for viewing and editing external database tables from within the program. You can open this dialog box using the following methods:

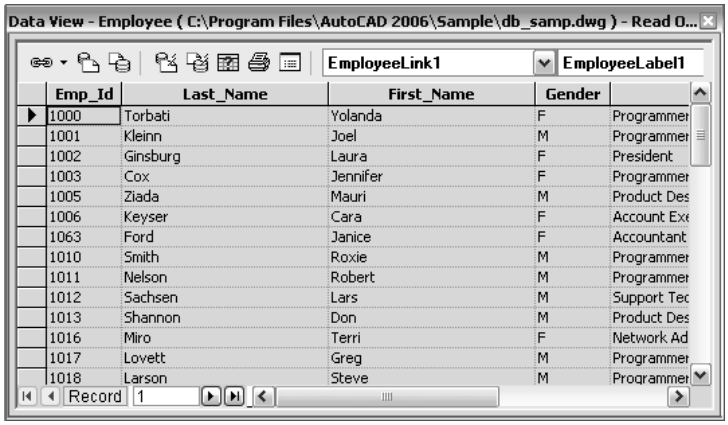
dbConnect Manager buttons ➤ Select a link template, a label template, or a table from the dbConnect Manager and choose View Table or Edit Table.



dbConnect ➤ View Data ➤ View External Table, Edit External Table, View Linked Table, or Edit Linked Table. (Displays a dialog box in which you can select a database table to open.)
Right-click a link template or a database table in the dbConnect Manager (page 332) and choose View Table or Edit Table.
Double-click a database table or a link template in the dbConnect Manager. Double-clicking a link template that has no links created in the drawing it's associated with opens the Link Template dialog box (page 372). The database table is opened in either Edit mode or Read-only mode, depending on the dbConnect Settings specified on the System tab of the Options dialog box. See *OPTIONS*.

The Data View window displays records from the selected database table. The Data View contains a set of buttons, a grid window for viewing and editing records, and a set of navigation controls for navigating through the record set. After you open the Data View window, the Data View menu is added to the menu bar. You can specify whether the Data View window can be docked or anchored by right-clicking the title bar and choosing an option from the shortcut menu.

The appearance of the Data View window is governed by your computer's system settings, and it can be changed in the Windows Control Panel. Text in the cells is displayed using the system window text color. If the database table is opened in Read-only mode, the cell background is shown using the system 3D objects button shade color (light gray by default). If it is opened in Edit mode, the cell background is shown using the system window color (white by default). When a cell is selected, it is shown using the system-selected items background and text colors.




### Data View Buttons

Create links and display linked records and graphical objects.

### Link

Links the currently selected database table row or rows to one or more graphical objects. The link and, if desired, the label from the currently selected link and label templates are created in the Data View window. To change the current link creation setting, choose the Link and Label Settings button. Three distinct link methods are available:

**Link** Creates a link to one or more graphical objects without creating a corresponding label. If the current drawing has a selection set already established, a link is created for each object in the selection set. If the current drawing does not have a current selection set, you are prompted to select objects to link to.

**Create Freestanding Label** Creates a freestanding label that isn't associated with a graphical object. You are prompted to specify an insertion point for the label.

**Create Attached Label** Creates a link to one or more graphical objects and creates a corresponding label. If the current drawing has a selection set already established, a link is created for all objects in the selection set. If the current drawing does not have a current selection set, you are prompted to select objects to link to.

### View Linked Objects in Drawing

Selects graphical objects in the current drawing that are linked to the currently selected Data View row or rows.

### View Linked Records in Table

Selects records in the Data View window that are linked to the current selection set of graphical objects.

### AutoView Linked Objects in Drawing

Displays linked objects automatically in the current drawing as you select rows from the database table.

### AutoView Linked Records in Table

Displays linked records automatically in the Data View window as you select graphical objects in the current drawing.

### Print Data View

Prints the contents of the Data View window to the current Microsoft® Windows® system printer.

### Data View and Query Options

Opens the Data View and Query Options dialog box (page 354), in which you can specify a number of settings that affect the interaction and display of linked objects in the Data View window and the current drawing.

### Query, Return to Query, and Return to Link Select

Opens either the New Query dialog box (page 376), the Query Editor (page 346), or the Link Select dialog box (page 369), depending on the method used to open the Data View window. If the Data View window was opened to view or edit a database table, this button's tooltip is Query, and choosing it opens the New Query dialog box. If the Data View window was opened to return the results of a query, this button's tooltip is Return to Query, and choosing it returns you to the Query Editor. If the Data View window was opened to return the results of a Link Select operation, this button's tooltip is Return to Link Select, and choosing it returns you to the Link Select dialog box.

### Link Template List

Creates a new link template, or specifies a link template from those that are defined for the open table. The selected link template is applied when you create new links in the current drawing. To create a new link template, select the New Link Template option from the Link Template list, and then choose the Link button flyout.

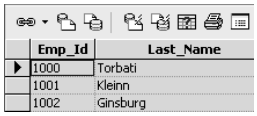
### Label Template List

Creates a new label template, or specifies a label template from those that are defined for the currently selected table. The selected label template is applied when you create new labels in the current drawing. To create a new label template, select the New Label Template option from the Label Template list and choose either the Create Freestanding Label or the Create Attached Label button flyout. If there is no link template defined for the selected database table in the current drawing, the New Link Template dialog box (page 375) is displayed.

### Data View Grid Window

Displays a subset of the records from a database table. You can select records by clicking the following elements:

- *Column header:* Selects all records in that column. Double-clicking a column header sorts its records in ascending order. Double-clicking a column header a second time sorts its records in descending order.
- *Record header:* Selects an individual record. Double-clicking a record header selects any graphical objects that the record is linked to in the drawing area.
- *Grid cell:* Selects one field of a given record. Double-clicking a cell selects it for editing.
- *Grid header:* Selects the entire table. Double-clicking the grid header commits any changes made during an editing session and closes the Data View window.



### Column Shortcut Menu

Displays the following options when you right-click one or more selected column headers.

**Sort** Opens the Sort dialog box (page 379), in which you can select a combination of up to five columns to use in specifying a sort order for the Data View.

A small triangle in the column header indicates that the data is sorted by that column.

**Hide** Removes all selected columns from the Data View display.

**Unhide All** Restores all hidden columns to the Data View display. Available only if you have hidden one or more columns.

**Freeze** Freezes all selected columns so that they do not scroll when you use the horizontal scroll bar. Available only if the selected columns are contiguous.

**Unfreeze All** Unfreezes all frozen columns so that they scroll when you use the horizontal scroll bar. Available only if you have frozen one or more columns.

**Align** Aligns the current column. *Standard* right-aligns numeric fields and left-aligns all others, *Left* left-aligns the column cells, *Center* center-aligns the cells, and *Right* right-aligns the cells.

**Find** Opens the Find dialog box (page 358), which you can use to search for a specific value. Find is limited to the values stored in the currently selected column.

**Replace** Opens the Replace dialog box (page 377), which you can use to search for a specific value to overwrite with a replacement value that you specify. Replace is limited to the values stored in the currently selected column. Available only for database tables that are opened in Edit mode.

### Cell Shortcut Menu

Displays the following options when you right-click the selected cell.

**View Linked Objects** Indicates graphical objects in the current drawing that are linked to the selected database record.

**Link** Links the current row to a graphical object. The link and, if desired, the label from the currently selected link and label templates are created in the Data View. You can specify whether a link, a freestanding label, or an attached

label is created by changing the settings from the Link and Label Settings cell shortcut menu option.

**Link and Label Settings** Lists the currently selected link creation mode. You can specify whether a link, a freestanding label, or an attached label is created when the Link shortcut menu option is chosen.

**Find** Opens the Find dialog box (page 358), which you can use to search for a specific value. The Find shortcut menu option limits its search to records contained in the same column as the currently selected cell.

**Replace** Opens the Replace dialog box (page 377), which you can use to search for a specific value to overwrite with a replacement value that you specify. Replace limits its search to records contained in the same column as the currently selected cell. Available only for database tables that are opened in Edit mode.

**Edit** Enables you to change the value of the currently selected cell. Available only for database tables that are opened in Edit mode.

**Cut** Clears the current cell and copies its value to the Clipboard. Available only for database tables that are opened in Edit mode.

**Copy** Copies the value from the current cell to the Clipboard.

**Paste** Inserts the value currently stored on the Clipboard into the selected cell. Available only for database tables that are opened in Edit mode.

**Clear** Deletes the value in the current cell. Available only for database tables that are opened in Edit mode.

### Record Shortcut Menu

Displays the following options when you right-click one or more selected record headers.

**View Linked Objects** Indicates graphical objects in the current drawing that are linked to the selected database records.

**Link** Links the selected row or rows to one or more graphical objects. The link and, if desired, the label from the currently selected link and label templates are created in the Data View window. You can specify whether a link, a freestanding label, or an attached label is created by choosing the Link and Label Settings shortcut menu option.

**Link and Label Settings** Lists the currently selected link creation mode. You can specify whether a link, a freestanding label, or an attached label is created when the Link shortcut menu option is chosen.

**Copy** Copies the selected records to the Clipboard.

**Delete Record** Deletes the selected records. Available only for database tables that are opened in Edit mode.

**Add New Record** Adds a new record with blank field values to the end of the record set. Available only for database tables that are opened in Edit mode.

**Clear All Marks** Clears record selection marks from the selected records in the Data View window.

### Grid Header Shortcut Menu

Displays the following options when you right-click the grid header.

**Commit** Saves all changes made in the Data View window to the source database table and closes the Data View window. Available only when a database table is open in Edit mode and you have edited its values.

**Restore** Undoes any changes made to a database table during an editing session and closes the Data View window. Available only when a database table is open in Edit mode and you have edited its values.

**Unhide All Columns** Restores any hidden columns to the Data View window display.

**Unfreeze All Columns** Unfreezes any frozen columns so that they scroll when you use the horizontal scroll bar.

**Clear All Marks** Clears all record selection marks from the Data View window.

**Print Preview** Opens the Print Preview window in the Data View window, in which you can preview your printed report.

**Print** Prints the contents of the Data View window to the current Windows system printer.

**Format** Opens the Format dialog box (page 359), in which you can control how data is displayed in the Data View window.

### Navigation Controls

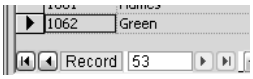
Cycle quickly through the records in the Data View window. The following controls are available:

**First Button** Selects the first record.

**Previous Button** Selects the previous record.

**Next Button** Selects the next record.

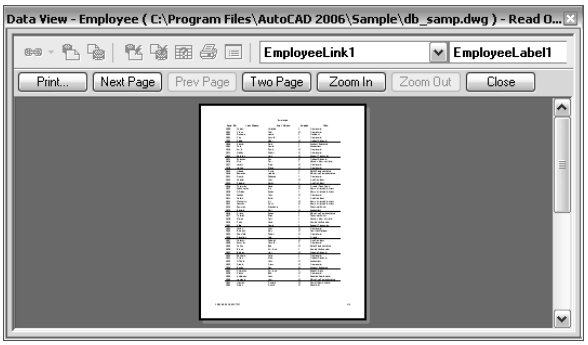
**Last Button** Selects the last record.




### Data View Print Preview Window

Loads into the Data View window a preview image of how the current table appears when you print it. When the Data View window is in Preview mode, none of the primary Data View buttons is available and the navigation buttons are removed from the bottom of the window. You can open this dialog box using the following methods:

Data View ➤ Print Preview
Right-click the grid header in the Data View window and choose Print Preview.




**Print** Opens the default system Print dialog box.

**Next Page** Presents a preview image of the next page of the database table.

**Prev Page** Restores the preview image of the previous page of the database table.

**Two Page** Toggles the display between one and two preview pages. If you choose Two Page, the name of the button changes to One Page and vice versa.

**Zoom In** Magnifies the preview page so that you can view the details more closely.

**Zoom Out** Shrinks the preview page to display a larger region of the database table.

**Close** Closes the Data View Print Preview window and restores the default Data View window display.

# Query Editor

## Quick Reference

Consists of a series of four tabs that you can use to build and execute queries. If you are creating a new query, the New Query dialog box (page 376) is displayed first. You can open the Query Editor using the following methods:

dbConnect buttons ➤ Select a database table or a link template in the dbConnect Manager and choose the New Query button.

Data View buttons ➤ Choose the Query or Return to Query button in the Data View window.

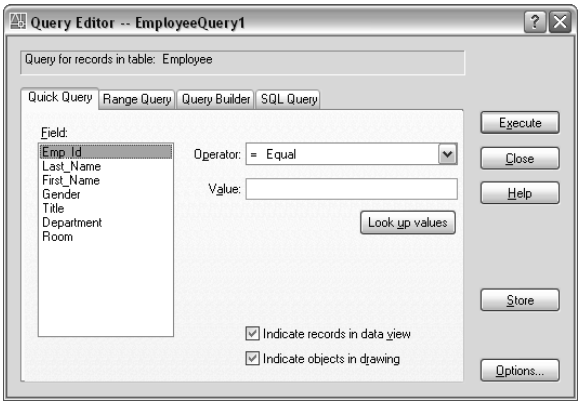
dbConnect ➤ Queries ➤ New Query on an External Table, New Query on a Link Template, or Edit Query. (Displays a dialog box in which you can select a database object to query.)
Right-click a database table or link template in the dbConnect Manager and choose New Query, or right-click a query and choose Edit.

- Quick Query (page 347)
- Range Query (page 348)
- Query Builder (page 349)
- SQL Queries (page 352)

# Quick Query Tab (Query Editor)

## Quick Reference

Develops simple queries based on a single database field, single operator, and single value.




**Field** Lists the fields from the current database table, from which you can select one to apply to the query.

**Operator** Displays a list of available operators that can be applied to the query. For information about operators, see "Construct Simple Queries" in the *User's Guide*.

**Value** Specifies a value for the field that you are using to construct your query.

**Look Up Values** Returns a list of all values for the specified field from the database table in the Column Values dialog box (page 353), from which you can select the value you want.

**Indicate Records in Data View** Indicates records that match your search criterion in the Data View window (page 337).

**Indicate Objects in Drawing** Indicates linked objects that match your search criterion in the current drawing.

**Execute** Issues the finished query and closes the dialog box.

**Close** Closes the dialog box without issuing the query.

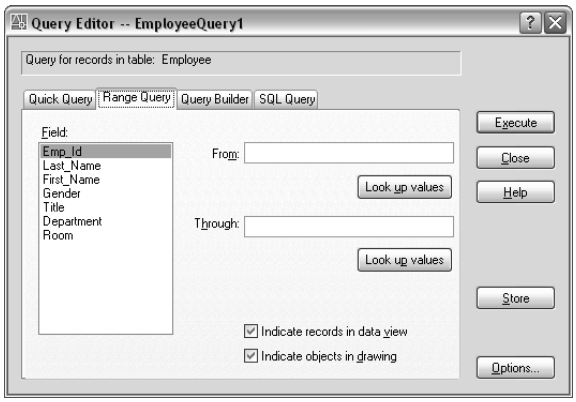
**Store** Saves the query with the current drawing.

**Options** Opens the Data View and Query Options dialog box (page 354).

# Range Query Tab (Query Editor)

## Quick Reference

Constructs a query that returns all records or objects that fall within a given range of values.




**Field** Lists the fields from the current database table, from which you can select one to apply to the query.

**From** Specifies the first value of the range. The query returns all records or graphical objects that are greater than or equal to this value.

**Look Up Values (From, Through)** Returns a list of all values for the specified field from the database table in the Column Values dialog box (page 353), from which you can select the value you want.

**Through** Specifies the second value of the range. The query returns all records or graphical objects that are less than or equal to this value.

**Indicate Records in Data View** Indicates records that match your search criterion in the Data View window (page 337).

**Indicate Objects in Drawing** Indicates linked objects that match your search criterion in the current drawing.

**Execute** Issues the finished query and closes the dialog box.

**Close** Closes the dialog box without issuing the query.

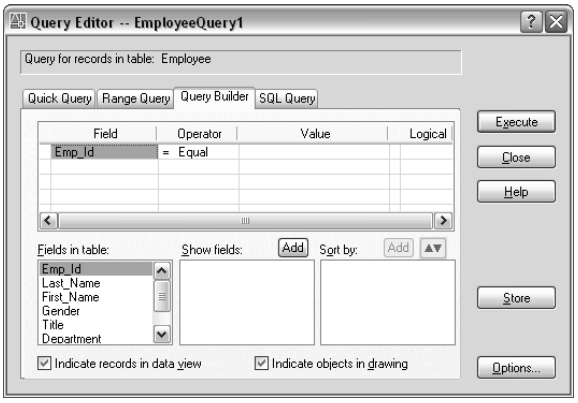
**Store** Saves the query with the current drawing.

**Options** Opens the Data View and Query Options dialog box (page 354).

# Query Builder Tab (Query Editor)

## Quick Reference

Constructs queries based on multiple search criteria. You can also group criteria parenthetically, select which fields to display in the returned query, and specify a sort order for the fields.




### Query Builder Grid

Provides a space for you to construct queries based on multiple search parameters.

**Parenthetical Grouping** Groups a series of search criteria by bracketing them within parentheses. You can nest up to four sets of parentheses within a single statement. To insert a beginning parenthesis, click in the cell to the left of the first Field cell that you want to group. To insert an end parenthesis, click in the cell to the right of the last Value cell that you want to group.

**Field** Provides a space where you can select the field or fields to include in your query. Double-click in the Field cell of the current row to display a list of available fields from the current database table that you can use in constructing your query. To add an additional parameter, specify an operator and a value for the current row and then select the Logical cell.

**Operator** Provides a space where you can select an operator to apply to the query condition of the current row. Double-click in the Operator cell to display a list of operators that you can use in constructing your query.

**Value** Provides a space where you can specify a value for the query condition of the current row. Click in the Value cell and enter a value, or choose the [...] button to retrieve a list of available values for the selected field in the Column Values dialog box (page 353).

**Logical** Provides either an And or an Or operator to the query statement. Click in the Logical cell to add an And value. To change the value to Or, click in the Logical cell again.

### Fields in Table

Displays a list of available fields from the current database table, from which you can specify the fields to display in the Data View window (page 337) when the query is executed. If you don't specify any fields to display, the query displays all fields from the database table. Double-clicking in a field in this list or selecting a field and choosing Add (Show Fields) adds the field to the list of those that are displayed in the Data View window by the returned query. You can also drag fields to add them to the Show Fields and Sort By areas.

### Show Fields

Specifies the fields that are displayed in the Data View window (page 337) when the query is executed. To remove a field from this list, drag it from the list to any area on the Query Builder tab.

### Add (Show Fields)

Adds a field to include in the Data View window (page 337) display of the returned query. To add a field, select it in the Fields in Table list window and then choose Add.

### Sort By

Specifies a sort order for the returned query. The first field added to the Sort By list is the primary sort. To change the sort order for a field, drag the field to a new location in the Sort By list. By default, fields are added to the Sort By list in an *ascending* sort order. To apply a *descending* sort, select a field and choose the Ascending/Descending Sort button, or double-click in the field. To remove a field, drag it from the list to any area on the Query Builder tab or select the field and press DELETE.

### Add (Sort By)

Adds a field to the Sort By list. To add a sort field, select it in the Fields in Table list window and then choose Add. Repeat for additional fields that you want to apply to the sort.

### Ascending/Descending Sort

Reverses the sort order for the currently selected field. If an ascending sort order is currently applied, choosing this button reverses the sort to descending order, and vice versa.

### Indicate Records in Data View

Indicates records that match your search criteria in the Data View window (page 337).

### Indicate Objects in Drawing

Indicates linked objects that match your search criteria in the current drawing.

### Execute

Issues the finished query and closes the dialog box.

### Close

Closes the dialog box without issuing the query.

### Store

Saves the query with the current drawing.

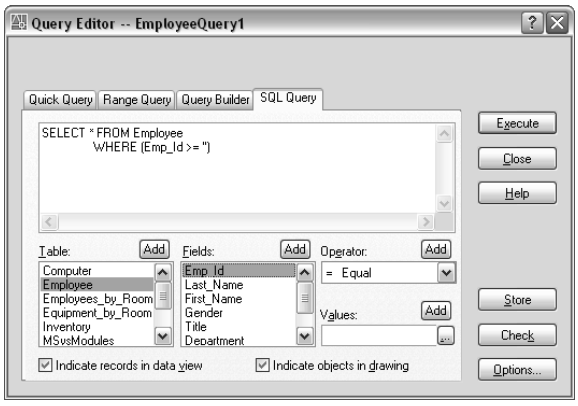
### Options

Opens the Data View and Query Options dialog box (page 354).

# SQL Query Tab (Query Editor)

## Quick Reference

Constructs any query statement that conforms with the SQL 92 protocol. The SQL Query tab provides both a query editor text box where you can type a free-form SQL statement and a set of tools to assist you in constructing your query. As not all database management systems are fully compliant with the SQL 92 standard, you should review your system's documentation to see what SQL commands are valid for your particular database.




**SQL Text Editor** Provides a space for you to type a free-form SQL query or add elements that you select using the various SQL Query tools.

**Table** Lists all database tables that are available in the current data source. You can add database tables to the SQL text editor by double-clicking them, by selecting them and choosing Add in the Table area, by dragging them from the Table list to the SQL text editor, or by entering their names directly in the SQL text editor.

**Add (Table)** Adds the currently selected database table to the SQL text editor.

**Fields** Displays a list of the fields in the selected database table. You can add fields to the SQL text editor by double-clicking them, by selecting them and choosing Add in the Fields area, or by dragging them from the Fields list to the SQL text editor.

**Add (Fields)** Adds the selected field to the SQL text editor.

**Operator** Displays a list of operators that you can apply to your query.

**Add (Operator)** Adds the selected operator to the SQL text editor.

**Values** Specifies a value for the selected field.

**Add (Values)** Adds the value specified in the Values area to the SQL text editor.

**[...] Button** Returns a list of available values for the specified field from the selected database table in the Column Values dialog box (page 353), from which you can select a value to apply to the query.

**Indicate Records in Data View** Indicates records that match your search criteria in the Data View window (page 337).

**Indicate Objects in Drawing** Indicates linked objects that match your search criteria in the current drawing.

**Execute** Issues the finished query and closes the dialog box.

**Close** Closes the dialog box without issuing the query.

**Store** Saves the query with the current drawing.

**Check** Checks your SQL query for proper syntax without actually executing it. This function helps you isolate syntax errors before you issue your query.

**Options** Opens the Data View and Query Options dialog box (page 354).

# Column Values Dialog Box

## Quick Reference

Lists values for the selected database column; you can select a value to apply to the current operation.

**Column Values** Lists all values for the selected database column. Select a value from the list and choose OK to apply it to the current operation. If you are constructing a query using the In operator, you can add additional values to the query by pressing the CTRL or SHIFT key as you select values.

## Configure a Data Source Dialog Box

### Quick Reference

Configures an external database so it can be accessed from the program. For detailed information about configuring a particular database system, refer to "Configure External Databases" in the *Driver and Peripheral Guide*. You can open this dialog box using the following methods:

dbConnect ➤ Data Sources ➤ Configure
Right-click the Data Sources node and choose Configure Data Source.
**Data Source Name** Specifies a name for the OLE DB data source that you are configuring.

**Data Sources** Lists all OLE DB data sources configured for use with the program that are present on your system.

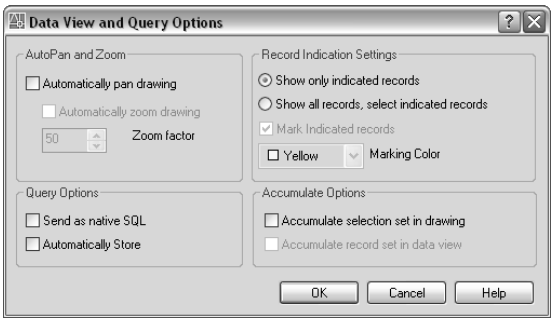
**OK** Opens the Microsoft Data Links Properties dialog box, in which you can continue configuring your data source.

## Data View and Query Options Dialog Box

### Quick Reference

Controls a number of settings related to the display of linked records and objects and the processing of SQL queries. You can open this dialog box using the following methods:

Data View buttons 
Data View ➤ Options. (The Data View menu is available only if you currently have a database table open in the Data View window.)




### AutoPan and Zoom

Control how linked objects are displayed in the current drawing when you select their corresponding records in the Data View window (page 337).

**Automatically Pan Drawing** Pans the drawing automatically to display objects that are associated with the current selection set of Data View records.

**Automatically Zoom Drawing** Zooms the drawing automatically so that all objects associated with the current record set are displayed.

**Zoom Factor** Specifies a zoom factor that limits the size of the extents of the indicated object set to a defined percentage of the drawing area. The available range is 20 to 90 percent with a default value of 50 percent. A value of 50 percent means that either the height of the extents is 50 percent of the height of the window, or the width of the extents is 50 percent of the window, whichever value is less.

### Record Indication Settings

Control the appearance of linked records in the Data View window (page 337) when their corresponding objects are selected in the current drawing.

**Show Only Indicated Records** Displays in the Data View window only the record set that is associated with the current selection set. Any records not linked to the current drawing selection set are not displayed.

**Show All Records, Select Indicated Records** Displays all records in the current database table. All records that are linked to the current selection set are selected in the Data View window.

**Mark Indicated Records** Applies a marking color to linked Data View records to clearly differentiate them from records without links.

**Marking Color** Specifies the marking color to apply to linked Data View records. The default color is yellow.

#### Query Options

Specify options for SQL query processing.

**Send as Native SQL** Issues queries to database tables in the format of the source table rather than SQL 92 format. You can use this option to issue proprietary commands in native database format.

**Automatically Store** Automatically stores queries when they are executed with the current drawing.

#### Accumulate Options

Control the accumulation of selection and record sets.

**Accumulate Selection Set in Drawing** Adds additional objects to the selection set as you select additional Data View records. If this option is cleared, the current drawing indicates a new set of objects each time you select a new set of Data View records.

**Accumulate Record Set in Data View** Adds additional records to the record set as you select additional graphical objects. If this option is cleared, the Data View window (page 337) indicates a new set of records each time you select a new set of graphical objects.

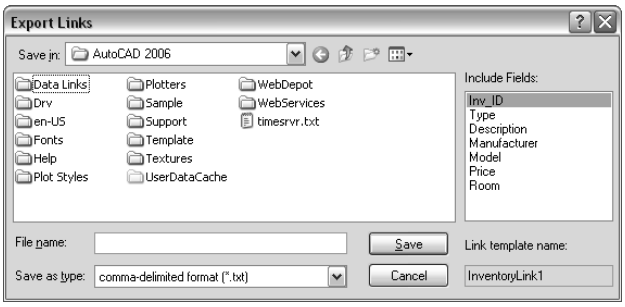
## Export Links Dialog Box

### Quick Reference

Exports a set of links that are associated with a selection set of graphical objects. You can open this dialog box using the following method:

dbConnect ➤ Links ➤ Export Links

If more than one link template exists for the set of graphical objects that you select, the Select a Database Object dialog box (page 378) is displayed, in which you can select a link template whose links you want to export.




**Include Fields** Specifies the database table fields to export. The key fields for the selected link template are included automatically. The handle of the object that each link is associated with is also exported.

**File Name** Specifies a name for the exported link file.

**Save as Type** Specifies the file format for the exported links. You can save the file in the native database format of the current database table or as a comma-delimited or space-delimited text file.

**Link Template Name** Displays the name of the link template whose links you're exporting.

# Export Query Set Dialog Box

## Quick Reference

Exports a set of queries that are associated with the current drawing. You can open this dialog box using the following methods:

dbConnect ➤ Queries ➤ Export Query Set
In the dbConnect Manager (page 332), right-click the drawing node of a drawing that contains one or more queries and choose Export Query Set.
**File Name** Specifies a name for the query set.

**Save as Type** Specifies the file format for the query set. Query sets are always saved with the *.dbq* file extension.

# Export Template Set Dialog Box

## Quick Reference

Exports a set of link templates and label templates that are associated with the current drawing. You can open this dialog box using the following methods:

dbConnect ➤ Templates ➤ Export Template Set
In the dbConnect Manager (page 332), right-click the drawing node of a drawing that has one or more templates defined and choose Export Template Set.
**File Name** Specifies a name for the template set.

**Save as Type** Specifies the file format for the template set. Template sets are always saved with the *.dbt* extension.

# Find Dialog Box

## Quick Reference

Searches for specified text or a numeric value in the currently loaded Data View table. The search is limited to a single table column. It is not possible to conduct a global search that scans all columns in the table. You can open this dialog box using the following methods:

Data View ➤ Find. (The Data View menu is available only if you currently have a database table open in the Data View window.)
Right-click a record header or a single cell in the Data View window and choose Find.
**Find What** Specifies the value to search for.

**Match Case** Searches for the exact value, including case, of what you enter in Find What. If this option is cleared, the program searches for the value regardless of case.

**Find Next** Finds the next occurrence of the value that you're searching for.

**Direction** Toggles the direction that the program searches for the specified value in the Data View window.

# Format Dialog Box

## Quick Reference

Controls how database table records are displayed in the Data View window (page 337). You can open this dialog box using the following methods:

Data View ➤ Format. (The Data View menu is available only if you currently have a database table open in the Data View window.)
Right-click the grid header in the Data View window and choose Format.
**Font** Specifies the font style that is applied to the current Data View table.

**Font Style** Specifies a style to apply to the current font. The available options are *Normal, Italic, Bold, Bold Italic, Not Bold,* and *Not Italic*.

**Size** Specifies the size in points that is applied to the current font.

**Effects** Applies additional formatting to the Data View window display. *Strikeout* draws a line through the center of all column data. *Underline* applies an underline to all column data. *Color* applies a color to all column data.

**Sample** Displays how the current Format option values look when applied to the Data View window.

**Script** Specifies the script that is applied to the current font.

# Import Query Set Dialog Box

## Quick Reference

Imports a set of queries into the current drawing. If the query set contains a name that matches a query in the current drawing, an alert box is displayed, and you can enter a different name for the query. You can open this dialog box using the following methods:

dbConnect ➤ Queries ➤ Import Query Set
Right-click a drawing node in the dbConnect Manager (page 332) and choose Import Query Set.
**Look In** Specifies the directory where the query set is located.

**File Name** Specifies the name of the query set to import.

**Files of Type** Specifies the file extension of the query set that you're importing. Query sets always have the *.dbq* extension.

## Import Template Set Dialog Box

### Quick Reference

Imports a set of link templates and label templates into the current drawing. If the template set contains a link or label template with a name that matches a template in the current drawing, an alert box is displayed where you can enter a different name for the template. You can open this dialog box using the following methods:

dbConnect ➤ Templates ➤ Import Template Set
Right-click a drawing node in the dbConnect Manager (page 332) and choose Import Template Set.

**Look In** Specifies the directory where the template set is located.

**File Name** Specifies the name of the template set to import.

**Files of Type** Specifies the file extension of the template set that you're importing. Template sets always have the *.dbt* extension.

## Label Template Dialog Box

### Quick Reference

Applies formatting to labels.

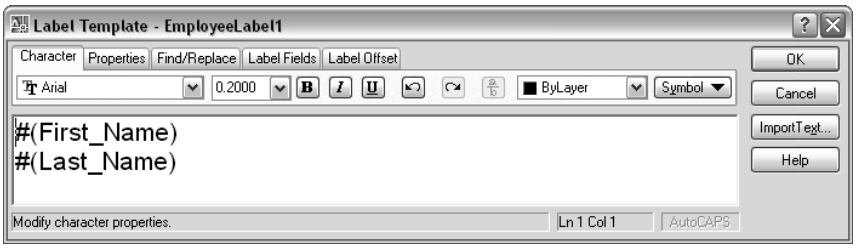
Before the Label Template dialog box opens, the New Label Template dialog box (page 375) is displayed, in which you can name the label template. You can open this dialog box using the following methods:

dbConnect Manager buttons ➤ Select a database table, link template, or label template and choose the New Label Template button. If a label template is selected, the program opens the Label Template dialog box without first displaying the New Label Template dialog box.

Data View buttons ➤ Choose New Label Template from the Label Template list, and then choose either the Create Freestanding Label or the Create Attached Label button flyout. If there is no link template defined for the selected database table in the current drawing, the program displays the New Link Template dialog box.
dbConnect ➤ Templates ➤ New Label Template. Select a link template in the Select a Database Object dialog box and choose Continue.
Right-click a link template or a database table in the dbConnect Manager and choose New Label Template.
Double-click a label template in the dbConnect Manager. This method opens the Label Template dialog box directly without first displaying the New Label Template dialog box.

### Character Tab

Controls character formatting for text that is entered at the keyboard or imported.




**Font** Specifies a font for new text or changes the font of selected text.

**Height** Sets the character height in drawing units for new text or changes the height of selected text. The default value is based on the current text style. If the current text style has no fixed height, the text height is the value stored in the *TEXTSIZE* system variable.

**Bold** Turns bold formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

**Italic** Turns italic formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

**Underline** Turns underlining on and off for new or selected text.

**Undo** Undoes actions, including changes to either text content or text formatting. You can also use CTRL+Z.

**Redo** Redoes actions, including changes to either text content or text formatting. You can also use CTRL+Y.

**Stack, Unstack** Creates stacked text, for example, fractions, if the selected text contains stack characters. Also, unstacks text if stacked text is selected. When the stack characters, carat (^), forward slash (/), and pound sign (#), are used, the text to the left of the stack character is stacked on top of the text to the right.

By default, text that contains a carat converts to left-justified tolerance values. Text that contains the forward slash converts to center-justified fractional numbers; the slash is converted to a horizontal bar the length of the longer text string. Text that contains the pound sign converts to a fraction separated by a diagonal bar the height of the two text strings. The characters above the diagonal fraction bar are bottom-right aligned; the characters beneath the diagonal bar are top-left aligned.

**Text Color** Specifies a color for new text or changes the color of selected text.

You can assign text the color associated with the layer it is on (BYLAYER) or the color of the block it is contained in (BYBLOCK). You can also select one of the colors in the color list or click Other to open the Select Color dialog box. (page 251)

**Symbol** Inserts a symbol or a nonbreaking space at the cursor position. Symbols can also be inserted manually. See Symbols and Special Characters (page 892).

Commonly used symbols are listed on the submenu. Click Other to display the Character Map dialog box, which contains the entire character set for each font available on your system. Select a character and click Select to place it in the Characters to Copy box. When you have selected all the characters that you want to use, click Copy and close the dialog box. In the Label Template dialog box, right-click and click Paste.

You can use the euro symbol with SHX fonts and their TrueType equivalent fonts shipped with AutoCAD 2000 and later releases. If your keyboard does not contain a euro symbol, hold down the ALT key and enter 0128 on the numeric keypad.

Symbols are not supported in vertical text.

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**NOTE** In the dialog box, the diameter symbol is displayed as **%%c**, but it is displayed correctly in the label. The nonbreaking space is unavailable in double-byte operating systems.

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### Properties Tab

Controls properties that apply to the whole label template.


**Style** Applies a text style to the label. The current style is saved in the *TEXTSTYLE* system variable. Character formatting for font, height, and bold or italic attributes is overridden if you apply a new style to an existing label template. Stacking, underlining, and color attributes are retained in characters to which a new style is applied.Styles that have backward or upside-down effects are not applied. If a style defined with a vertical effect is applied to an SHX font, the text is displayed horizontally in the Label Template dialog box.

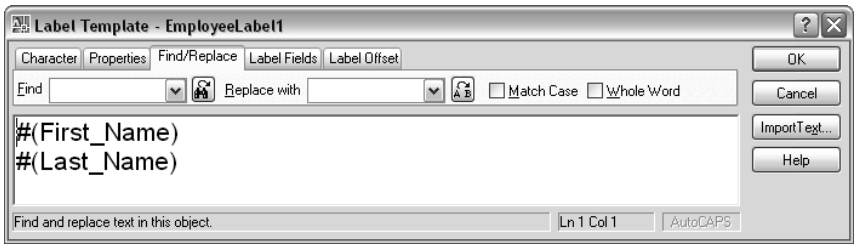
**Justification** Sets justification and alignment for the label. Top Left is the default setting. Spaces entered at the end of a line are included as part of the text and affect the justification of the line. Text is center-, left-, or right-justified with respect to the left and right text boundaries. Text is middle-, top-, or bottom-aligned with respect to the top and bottom text boundaries. See -MTEXT (page 888) for an illustration of the nine justification options.

**Width** Sets a paragraph width for new or selected text. The No Wrap option produces a single line. The width of individual characters is not affected by this option.

**Rotation** Sets the rotation angle for the label in the current unit of angle measurement (degrees, radians, or grads).

### Find/Replace Tab

Searches for specified text strings in the label template and replaces them with new text.




**Find** Defines the text string to search for.

**Find Button** Starts a search for the text string in Find. To continue the search, click the Find button again.

**Replace With** Defines the text string to replace the text in Find What.

**Replace Button** Replaces the highlighted text with the text in Replace With.

**Match Case** Finds text only if the case of all characters in the text string is identical to the case of the text in Find. When this option is cleared, a match is found for specified text strings regardless of case.

**Whole Word** Matches the text in the Find box only if it is a single word. Text that is part of another word is ignored. When this option is cleared, a match is found for text strings, whether they are single words or parts of other words.

### Label Fields Tab

Specifies which database table fields are displayed when you create labels based on this template.

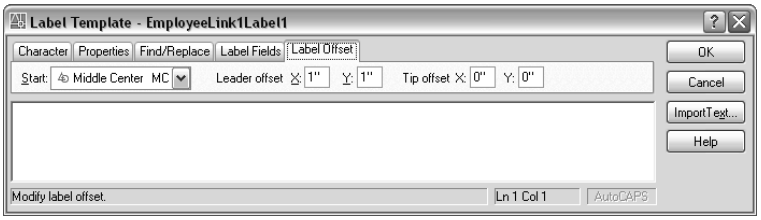

**Field** Displays a list of available fields from the current database table that you can include in your label. You may include any number of fields from the source table.

**Add** Adds the selected field in the Field list to the label.

**Table** Displays the name of the current database table.

### Label Offset Tab

Defines *X* and *Y* coordinate offsets for the insertion point of both your label and, if it is an attached label, its associated leader object.




**Start** Specifies the start point for the leader object. The point is defined with respect to the extents of the graphical object that the label is attached to. The Start option has no effect on freestanding labels, as they have no associated leader object.

**Leader Offset** Specifies an *X* and *Y* offset for the multiline text object with respect to the associated leader object. By default, the X and Y Leader Offset options are set to 1, indicating that the label text is offset 1 unit in the *X* and *Y* planes from the tip of the leader object. The Leader Offset settings apply only to attached labels.

**Tip Offset** Specifies the leader tip or label text offset values for attached and freestanding labels. For attached labels, the Tip Offset specifies the *X* and *Y* offset from the value specified in Start for the tip of the leader object. For freestanding labels, the Tip Offset specifies the *X* and *Y* offset from the insertion point you specified for the label.

## Label Template Properties Dialog Box

### Quick Reference

Displays the full data source path to a label template's database table. You can use this dialog box to update a label template with new data source information.

Before the Label Template Properties dialog box opens, the Select a Database Object dialog box is displayed, from which you can select a label template.

You can open this dialog box using the following method:

dbConnect ➤ Templates ➤ Label Template Properties

**Template Name** Displays the name of the selected label template.

**Data Source** Displays the name of the data source specified for the selected label template. You can select a new data source from this list to apply to the label template.

**Catalog** Displays the name of the catalog specified for the selected label template. You can select a new catalog from this list to apply to the label template.

**Schema** Displays the name of the schema specified for the selected label template. You can select a new schema from this list to apply to the label template.

**Table** Displays the name of the database table specified for the selected label template. You can select a new database table from this list to apply to the label template.

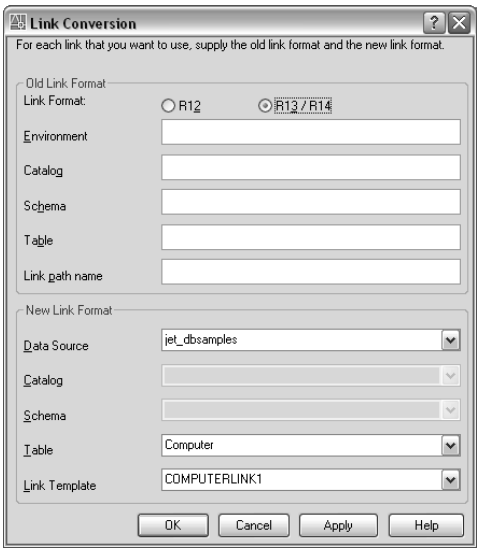
**Fields** Displays the fields specified for the selected label template.

## Link Conversion Dialog Box

### Quick Reference

Converts links created in AutoCAD Release 12, Release 13, and Release 14 to AutoCAD 2000 (and later) format. You can open this dialog box using the following method:

dbConnect ➤ Link Conversion




Because legacy links are stored in formats that differ significantly from AutoCAD 2000 and later, the link conversion process requires that you specify a mapping of the old AutoCAD SQL Extension™ (ASE) link values to the new ones used by AutoCAD 2000 and later. For example, you must identify the environment used by a particular link in Release 14 and specify the corresponding data source that you want substituted for this environment in AutoCAD 2000 and later. Similar mappings must be established between all old ASE link values and the new values that you want substituted for them.

### Link Format

Specifies the format of the link that you are converting. Different fields are displayed depending on whether you select R12 or R13/R14.

**R12** Specifies settings for converting R12 links.

**R13/R14** Specifies settings for converting R13 and R14 links.

### Old Link Format (R12)

Specifies the path to the database table used by the link that you're converting.

**DBMS** Specifies the database management system used by the link that you're converting.

**Database** Specifies the database used by the link that you're converting.

**Table** Specifies the database table used by the link that you're converting.

### Old Link Format (R13/R14)

Specifies the path to the database table used by the link that you're converting.

**Environment** Specifies the environment used by the link that you're converting.

**Catalog** Specifies the catalog used by the link that you're converting.

**Schema** Specifies the schema used by the link that you're converting.

**Table** Specifies the database table used by the link that you're converting.

**Link Path Name** Specifies the link path name used by the link that you're converting.

### New Link Format

Specifies the path to the database table used by the converted link.

**Data Source** Lists available data sources that you can select for the converted link.

**Catalog** Lists available catalogs that you can select for the converted link.

**Schema** Lists available schemas that you can select for the converted link.

**Table** Lists available database tables that you can select for the converted link.

**Link Template** Lists available link templates that you can select for the converted link.

### OK

Writes conversion mapping information to the *asi.ini* file and closes the dialog box.

### Apply

Writes conversion mapping information to the *asi.ini* file but keeps the dialog box open so that you can specify conversions for additional links.

### Cancel

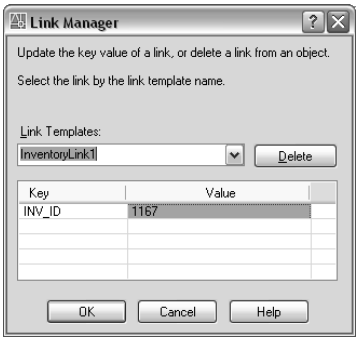
Closes the dialog box without converting any links.

# Link Manager

## Quick Reference

Edits the values of the key fields for a selected linked object. You can open the Link Manager using the following methods:

dbConnect ➤ Links ➤ Link Manager. Select a linked graphical object. Select and then right-click a linked graphical object, and then choose Link ➤ Link Manager.




**Link Templates** Lists all link templates that are associated with the selected graphical object. Select the link template whose link values you want to view or edit.

**Key** Displays the key fields of the currently selected link template.

**Value** Displays all key values that are defined for the selected link. To display a list of values for a specific field, click a Value cell and choose [...].

**Delete** Deletes the currently selected link from the graphical object it's associated with.

# Link Select Dialog Box

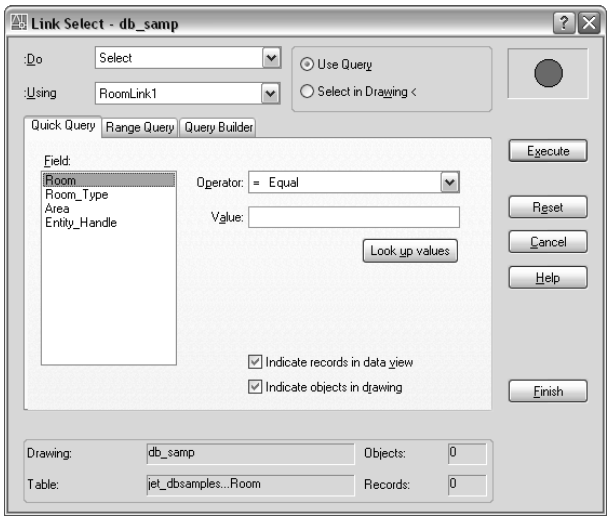
## Quick Reference

Constructs iterative selection sets of graphical objects and database records. The Link Select dialog box, known as ASE Select in previous releases, is an advanced implementation of the Query Editor. You begin a Link Select

operation by creating an initial query or selection set of graphical objects (set A), which can be operated on by an additional query or selection set (set B). The results of the returned operation become the new running selection set (set A), and you can apply additional queries or selection sets to further refine your running query.

In order to work with the Link Select dialog box, you must have a drawing open that has a number of links already created. You can open this dialog box using the following methods:

dbConnect ➤ Links ➤ Link Select
Right-click a link template in the dbConnect Manager (page 332) and choose Link Select.




The Link Select dialog box contains the Quick Query, Range Query, and Query Builder tabs from the Query Editor (page 346), and the following additional controls.

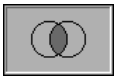
### Do

Applies a Link Select operator to the current query or selection set. The following operators are available: Select, Union, Intersect, Subtract A-B, Subtract A-B.

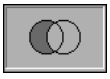
**Select** Creates an initial query or selection set. This selection set can be refined through subsequent Link Select operations.

**Union** Adds the results of the new query or selection set to the running selection set. This operation returns all records or objects that are members of set A *or* set B.

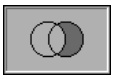
**Intersect** Returns the intersection of the existing running selection and the results of the new query or drawing selection. This operation returns all records or objects that are members of set A *and* set B.



**Subtract A-B** Subtracts the results of the new query or drawing selection from the existing running selection.



**Subtract B-A** Subtracts the existing running selection from the results of the new query or drawing selection.



### Using

Lists available link templates that you can apply to the current selection set.

### Use Query

Uses any of the available Query Editor tabs to construct a query that returns a selection set to the Link Select operation.

### Select in Drawing

Closes the Link Select dialog box when you choose the Select button so that you can construct a selection set of graphical objects to return to the Link Select operation.

#### Execute/Select

Executes the current Link Select query or temporarily dismisses the Link Select dialog box so you can select graphical objects from the drawing. If the Use Query option is selected, the Execute button is displayed. If the Select in Drawing option is selected, the Select button is displayed. After choosing the Execute or Select button, you can apply more queries or select additional graphical objects to further refine your selection set.

#### Venn Diagram

Displays a diagram of the currently selected Do operation.

#### Reset

Clears the Link Select dialog box and discards all previous actions so that you can begin a new Link Select operation.

#### Cancel

Closes the dialog box and discards all actions.

#### Finish

Completes the Link Select operation and closes the dialog box. Returns either a selection set of graphical objects, a subset of Data View records, or both, depending on the parameters specified in the Link Select operation.

#### Status Bar

Displays the current status information based on the running Link Select operation. The current drawing and table are displayed, as well as the number of linked objects and records that currently meet the specified Link Select parameters.

## Link Template Dialog Box

### Quick Reference

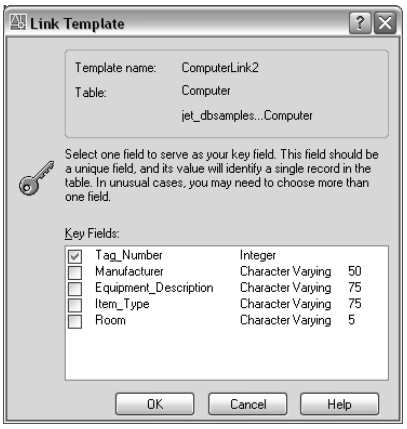
Specifies the key fields that are used by a link template.

Before the Link Template dialog box opens, the New Link Template dialog box (page 375) is displayed, in which you can name the link template. You can open this dialog box using the following methods:

dbConnect Manager buttons ➤ Select a database table and choose the New Link Template button.

Data View buttons ➤ Choose New Link Template from the Link Template list, and then choose the Link button flyout.

dbConnect ➤ Templates ➤ New Link Template. Navigate to and select a database table from the Select a Database Object dialog box and choose Continue.
Right-click a database table in the dbConnect Manager and choose New Link Template.
Double-click a link template in the dbConnect Manager. This method opens the Link Template dialog box directly, without first displaying the New Link Template dialog box.




**Template Name** Displays the name of the link template.

**Table** Displays the current database table. The first line lists the name of the table, and the second line lists the full data source path to the table in the following syntax: *environment.catalog.schema.table*

**Key Fields** Displays all fields that are available for the current database table, as well as the field type and, in the case of character fields, the field length. You specify the link template's key fields by selecting the check box to the left of the field name.

# Link Template Properties Dialog Box

## Quick Reference

Displays the full data source path to a link template's table. You can use this dialog box to update a link template with new data source information.

Before the Link Template Properties dialog box opens, the Select a Database Object dialog box (page 378) is displayed, from which you can select a link template.

You can open this dialog box using the following method:

dbConnect ➤ Templates ➤ Link Template Properties

**Template Name** Displays the name of the selected link template.

**Data Source** Displays the name of the data source specified for the selected link template. You can select a new data source from this list to apply to the link template.

**Catalog** Displays the name of the catalog specified for the selected link template. You can select a new catalog from this list to apply to the link template.

**Schema** Displays the name of the schema specified for the selected link template. You can select a new schema from this list to apply to the link template.

**Table** Displays the name of the database table specified for the selected link template. You can select a new table from this list to apply to the link template.

**Keys** Displays the key fields specified for the selected link template.

# New Label Template Dialog Box

## Quick Reference

Creates a new label template. You can open this dialog box using the following methods:

dbConnect Manager buttons ➤ Select a database table and choose the New Label Template button.

Data View buttons ➤ Choose New Label Template from the Label Template list, and then choose either the Create Freestanding Label or the Create Attached Label button flyout. If there is no link template defined for the selected database table in the current drawing, the program displays the New Link Template dialog box.
dbConnect ➤ Templates ➤ New Label Template. Select a link template from the Select a Database Object dialog box and choose Continue.
Right-click a database table in the dbConnect Manager and choose New Label Template.

After you enter a name for the label template and choose Continue, the Label Template dialog box (page 360) opens, in which you specify the fields to display in your label and how to format them.

**New Label Template Name** Specifies a name for the label template.

**Start with Template** Lists available label templates in the current drawing, from which you can select one to use as a start point for your new label template.

# New Link Template Dialog Box

## Quick Reference

Creates a new link template. You can open this dialog box using the following methods:

dbConnect Manager buttons ➤ Select a database table and choose the New Link Template button.

Data View buttons ➤ Choose New Link Template from the Link Template list, and then choose the Link button flyout.

dbConnect ➤ Templates ➤ New Link Template. Navigate to and then select a database table in the Select a Data Object dialog box and choose Continue. Right-click a table in the dbConnect Manager and choose New Link Template.

After you enter a name for the link template and choose Continue, the Link Template dialog box (page 372) opens, in which you specify the key fields that the link uses.

**New Link Template Name** Specifies a name for the link template.

**Start with Template** Lists all available link templates from the current drawing, from which you can select one to use as a starting point for the new link template.

## New Query Dialog Box

### Quick Reference

Creates a new query and opens the Query Editor. You can open this dialog box using the following methods:

dbConnect Manager buttons ➤ Select a link template, database table, or existing query and choose the New Query button.

dbConnect ➤ Queries ➤ New Query on an External Table or New Query on a Link Template. (A dialog box is displayed in which you can select a database object to query.)

Right-click a link template, database table, or existing query in the dbConnect Manager and choose New Query.

After you enter a name for the query and choose Continue, the Query Editor (page 346) is displayed, in which you can construct a new query.

**Query for Records in Table** Read-only field that indicates the table to query.

**New Query Name** Specifies a name for the query.

**Existing Query Names** Lists all available queries from the current drawing.

# Replace Dialog Box

## Quick Reference

Searches for specified text or a numeric value in the currently loaded Data View table. You can specify a replacement value that overwrites the value that you're searching for. The search is limited to a single database table column. It is not possible to conduct a global search that scans all columns in the database table. You can open this dialog box using the following methods:

Data View ➤ Replace. (This menu is available only if you have a database table open in the Data View window.)
Right-click a single cell or column header in the Data View window and choose Replace.

**Find What** Specifies the value to search for.

**Replace With** Specifies a replacement value for the value that you're searching for.

**Match Case** Searches for the exact value, including case, of what you enter in Find What. If this option is cleared, the program searches for the value regardless of case.

**Find Next** Finds the next occurrence of the value that you're searching for.

**Replace** Replaces the next occurrence of the value that you're searching for with the value specified in Replace With.

**Replace All** Replaces all occurrences of the value that you're searching for with the value specified in Replace With.

## Select a Database Object Dialog Box

### Quick Reference

Displays a list of the database objects, such as link templates, label templates, or queries, that are associated with the current drawing. You can select a database object from this list to apply to the current operation. You can open this dialog box using the following methods:

dbConnect ➤ Templates ➤ New Label Template, Edit Link Template, Edit Label Template, Delete Link Template, or Delete Label Template; Queries ➤ Execute Query, New Query on a Link Template, Edit Query, or Delete Query; Links ➤ Delete Links; Labels ➤ Reload Labels, Show Labels, Hide Labels, or Delete Labels; View Data ➤ View Linked Table, Edit Linked Table

**Database Object List** Lists the available database objects that are associated with the current drawing, from which you must select one to apply to the current operation.

**OK/Continue** Either completes the current operation or opens a secondary dialog box in which you can continue the current operation.

If, for example, you are creating a new label template, the New Label Template dialog box is displayed.

## Select Data Object Dialog Box

### Quick Reference

Displays the Data Sources node of the dbConnect Manager (page 332), from which you can navigate to and select a database table that the current operation is applied to. You can open this dialog box using the following methods:

dbConnect ➤ Data Sources ➤ Connect; Templates ➤ New Link Template; Queries ➤ New Query on an External Table; View Data ➤ View External Table or Edit External Table

**Tree View Window** Lists the available data sources, from which you can select a database table to apply to the current operation.

**OK/Continue** Either completes the current operation or opens a secondary dialog box or window, in which you can continue the current operation.

If, for example, you are creating a new link template, the New Link Template dialog box is displayed.

# Sort Dialog Box

## Quick Reference

Specifies a sort order that is applied to the database table records displayed in the Data View window (page 337). You can select a combination of up to five columns to apply to the sort order. You can open this dialog box using the following method:

Right-click a column header in the Data View window and choose Sort.
**Sort By** Specifies the primary column to apply to the sort operation. The data from this column is sorted first, and then the data from any additional Then By columns that are specified is sorted.

**Ascending** Applies an ascending sort order to the selected column. The column data is ordered with the smallest value at the beginning of the Data View record set and the largest value at the end of the record set.

**Descending** Applies a descending sort order to the selected column. The column data is ordered with the largest value at the beginning of the Data View record set and the smallest value at the end of the record set.

**Then By** Specifies an additional column to apply to the sort operation.

**Reset** Clears all columns that are specified for the current sort order so that you can specify a new sort order.

# Synchronize Dialog Box

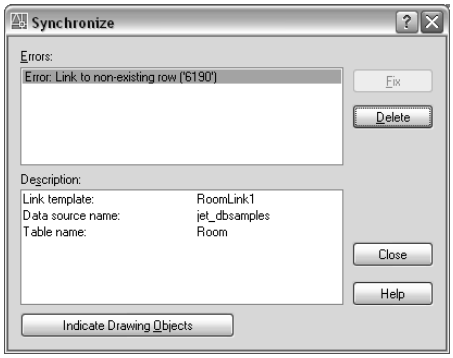
## Quick Reference

Updates links that the program is unable to resolve. Links can become invalid if you change the underlying structure of their source database table, or if you move the source table to a new location. In these cases, the structure specified by the link template used to create the links is no longer valid, and link templates and any associated links must be updated.

The Synchronize dialog box provides a list of detected errors. Certain errors (such as a resized column in the source database table) can be fixed directly from the Synchronize dialog box. Other errors (such as links that point to nonexistent records) must be fixed in the source database table.

You can open this dialog box using the following methods:

dbConnect ➤ Synchronize
Right-click a link template in the dbConnect Manager (page 332) and choose Synchronize.




**Errors** Displays a list of errors detected. You can select an error to fix.

**Description** Provides a brief description of the error selected in the Errors list.

**Fix** Fixes the currently selected error.

**Delete** Deletes the invalid link template and any associated links from the current drawing.

**Close** Closes the dialog box.

**Indicate Drawing Objects** Indicates all objects in the current drawing that are associated with the invalid link template.

# DBLIST

## Quick Reference

Lists database information for each object in the drawing

dblist

The text window displays information about each object in the current drawing. The program pauses when the window fills with information. Press ENTER to resume output, or press ESC to cancel.

# DDEDIT

## Quick Reference

Edits single-line text, dimension text, attribute definitions, and feature control frames

Text 

Modify ➤ Object ➤ Text ➤ EditAt the Command prompt, enter ddedit.
Double-click a text object.
Select a text object, right-click in the drawing area, and click Edit.
**ddedit**
Select an annotation object or [Undo]:
**Object Selection** Displays the appropriate editing method for the type of text you select:

- When the DTEXTED system variable is set to 0 or 2, text created using *TEXT* or DTEXT displays the In-Place Text Editor (page 873) without the Text Formatting toolbar and the ruler. Right-click to display options (page 1430).
- Text created using *MTEXT* displays the In-Place Text Editor (page 873).
- Attribute definitions (not part of a block definition) display the Edit Attribute Definition dialog box (page 382).
- Feature control frames display the Geometric Tolerance dialog box (page 1437).

DDEDIT repeats the prompt until you press ENTER to end the command.

**Undo** Returns the text or attribute definition to its previous value. You can use this option immediately after editing.

## Edit Text Dialog Box

### Quick Reference

Text 
Select a text object, right-click in the drawing area, and click Edit.
**ddedit**

When the *DTEXTED* system variable is set to 1, modifies text created with *TEXT* or DTEXT. Enter the new text and click OK.

To add a field to the text, right-click the text where you want to insert the field and click Insert Field on the shortcut menu to display the Field dialog box (page 578).

## Edit Attribute Definition Dialog Box

### Quick Reference

Text 
Select a text object, right-click in the drawing area, and click Edit.
**ddedit**

Modifies an attribute definition's tag, prompt, and default value.

**Tag** Specifies the attribute tag, which identifies the attribute in the drawing. The tag can contain any characters except spaces or exclamation marks (!). Lowercase letters are automatically changed to uppercase.

**Prompt** Specifies the attribute prompt that is displayed when you insert a block containing this attribute definition. If you need leading blanks in the prompt, start the string with a backslash (\). If the first character must be a backslash, start the string with two backslashes.

**Default** Specifies the default attribute value. If you need leading blanks in the default value, start the string with a backslash (\). If the first character must be a backslash, start the string with two backslashes.

To use a field as the value, right-click and click Insert Field on the shortcut menu to display the Field dialog box (page 578).

# DDPTYPE

## Quick Reference

Specifies the display style and size of point objects

Format ➤ Point StyleAt the Command prompt, enter ddptype.
**ddptype** (or '**ddptype** for transparent use)

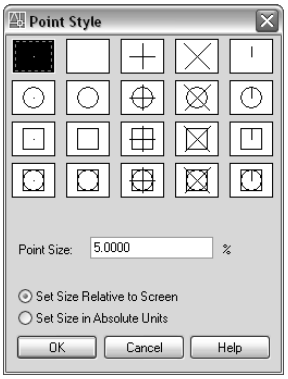
The Point Style dialog box (page 383) is displayed.

# Point Style Dialog Box

## Quick Reference

Format ➤ Point Style
**ddptype** (or '**ddptype** for transparent use)

Shows the current point style and size. Change the point style by selecting an icon.



#### Point Display Images

Specifies the image used to display point objects. The point style is stored in the *PDMODE* system variable.

#### Point Size

Sets the point display size. The value you enter can be relative to the screen or in absolute units. The point display size is stored in the *PDSIZE* system variable. Subsequent point objects that you draw use the new value.

**Set Size Relative to Screen** Sets the point display size as a percentage of the screen size. The point display does not change when you zoom in or out.

**Set Size in Absolute Units** Sets the point display size as the actual units you specify under Point Size. Points are displayed larger or smaller when you zoom in or out.

# DDVPOINT

### Quick Reference

Sets the three-dimensional viewing direction

View ➤ 3D Views ➤ Viewpoint PresetsAt the Command prompt, enter ddvpoint.
**ddvpoint**

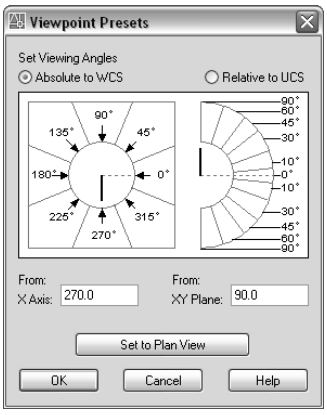
The Viewpoint Presets dialog box (page 384) is displayed.

## Viewpoint Presets Dialog Box

### Quick Reference

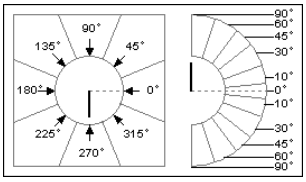
View ➤ 3D Views ➤ Viewpoint Presets
**ddvpoint**

Defines 3D view settings.




### Set Viewing Angles

Sets the direction of the view relative to either the world coordinate system (WCS) or a user coordinate system (UCS).




**Absolute to WCS** Sets the view direction relative to the WCS.

**Relative to UCS** Sets the view direction relative to the current UCS.

### From

Specifies viewing angles.

**X Axis** Specifies the angle from the *X* axis.

**XY Plane** Specifies the angle from the *XY* plane.

You can also use the sample image to specify viewing angles. The black arm indicates the new angle. The gray arm indicates the current angle. Specify an angle by selecting the inner region of the circle or half-circle. Selecting the bounded outer regions rounds off the angle to the value displayed in that region. If you select the inner arc or an area inside it, the angle is not rounded off, and the result may be a fractional number.

#### Set to Plan View

Sets the viewing angles to display the plan view (*XY* plane) relative to the selected coordinate system.

# DELAY

## Quick Reference

Provides a timed pause within a script

**delay** (or '**delay** for transparent use)
Enter delay time (in milliseconds): *Enter a value from 0 through 32,767 milliseconds*

Specifies the duration of a pause. Entering **delay 1000** in your script delays the start of execution of the next command for about one second. The longest delay available is 32767, which is slightly less than 33 seconds.

# DETACHURL

## Quick Reference

Removes hyperlinks in a drawing

**detachurl**
Select objects: *Use an object selection method, and press* ENTER *to end selection*

Hyperlinks are removed from the selected objects. If an area is selected, the polyline is deleted. You can use *PURGE* to remove the URLLAYER layer.

# DGNADJUST

## Quick Reference

Changes the display options of selected DGN underlays

**dgnadjust**
Select DGN underlay: *Select one or more DGN underlays*

Enter DGN underlay option [Fade (page 387)/Contrast (page 387)/Monochrome (page 387)] <Fade>:

**NOTE** The default values for Fade, Contrast, Monochrome remain as they were set the last time the command was used. To confirm the settings on a selected DGN underlay, use the Properties palette.

#### Fade

Controls the fade effect of the underlay. Values range from 0 through 80. The greater the value, the more transparent the underlay appears against the background. The lesser the value, the less transparent and closer to opaque the underlay appears. A value of 0 makes the underlay appear fully opaque.

Enter fade value (0-80): *Enter a value*

#### Contrast

Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color.

Enter contrast value (0-100): *Enter a value*

#### Monochrome

This Yes/No toggle controls the color saturation of all linework while maintaining the luminance. When turned on, the linework appears in varying shades of gray starting at black if the background color luminance is 50% or more. If the background color luminance is less than 50%, then the colors are inverted, with the darkest linework displaying in white, and the lightest linework displaying in black.

Monochrome? [Yes/No]: *Enter* **yes** *or* **no** *and then press ENTER*

# DGNATTACH

## Quick Reference

Attaches a DGN underlay to the current drawing

Insert 

Insert ➤ DGN UnderlayAt the Command prompt, enter dgnattach.

**dgnattach**

The Select DGN File dialog box (a standard file selection dialog box (page 931)) is displayed. Once you select a DGN file, the Attach DGN Underlay dialog box (page 388) is displayed.

If you enter **-dgnattach** at the Command prompt, options are displayed at the command prompt (page 390).

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**NOTE** When a DGN file is attached as an underlay, its levels structure (layers) are combined into a single layer. The DGN underlay is placed on the current layer. To hide the DGN attachment, freeze the layer on which it was attached.

---

# Attach DGN Underlay Dialog Box

## Quick Reference

Insert 

Insert ➤ DGN UnderlayAt the Command prompt, enter dgnattach.

dgnattach

Names, locates and defines the insertion point, scale and rotation of attached DGN underlays.

### Name

Identifies the DGN file you have selected to attach, either from the Select DGN File dialog box (an unattached DGN file) or from the list of previously attached DGN underlays. To add another instance of a DGN underlay that is already attached, select the DGN name from the list.

**Browse** Opens the Select DGN File dialog box (a standard file selection dialog box (page 931)).

**Found In** Displays the path where the DGN file was located.

**Saved Path** Displays the path that is saved with the drawing when the DGN file is attached. The path is dependent upon the Path Type setting.

### Select a Design Model from the DGN File

Displays all of the design models that are found in the DGN file. Sheet models in the DGN file are not listed. If the DGN file only contains a single design model, that model is listed.

If the DGN file contains multiple models, only a single model can be selected for attachment. The first model in the list is selected by default.

### Conversion Units

Select the appropriate conversion units for the DGN underlay. The DGN file contains working units (imperial or metric) called *master units* and *sub-units*. The selected working units (master units or sub-units) are converted one-for-one into DWG units.

**Master Units** Specifies that one master unit of the imported DGN file converts to one DWG drawing unit.

**Sub Units** Specifies that one sub-unit of the imported DGN file converts to one DWG drawing unit.

### Path Type

Specifies one of three types of folder path information to save with an attached DGN underlay: a full (absolute) path, a relative path, and no path.

**Full Path** Specifies the full path to the DGN file.

**Relative Path** Specifies a relative path to the DGN file.

**No Path** Specifies only the DGN file name. The DGN file should be located in the folder with the current drawing file.

### Insertion Point

Specifies the insertion point for the selected model in the DGN file.

**Specify On-Screen** Directs input at the command prompt or the pointing device.

**X** Sets the *X* coordinate value.

**Y** Sets the *Y* coordinate value.

**Z** Sets the *Z* coordinate value.

#### Scale

Specifies the scale factor of the selected image.

If INSUNITS (page 1727) is set to "unitless" or if the underlay does not contain information about its extents, the scale factor becomes the underlay width in AutoCAD units. If INSUNITS has a value such as millimeters, centimeters, inches, or feet, and the underlay has information about its extents, the scale factor is applied after the true width of the underlay in AutoCAD units is determined.

For best results, set INSUNITS to match the units used to create the DGN file.

#### Rotation

Specifies the rotation angle of the selected design model.

## -DGNATTACH

### Quick Reference

If you enter **-dgnattach** at the command prompt, the following DGNATTACH command prompts are displayed.

Path to DGN file (page 391) to attach: *Enter a path and file name, or enter ~ to display a file selection dialog box*
Enter name of model or [? (page 391)] <*name of design model*>: *Enter the name of a design model in the DGN file, enter* **?** *to list all design models, or press* ENTER
Specify conversion units [Master/Sub] <Master>: *Specify* **m** *or* **s**, *or press* ENTER
Specify insertion point: *Specify a point*
Specify scale factor or [Unit (page 391)] <1.0000>: *Enter a value, enter* **u**, *or press* ENTER
Specify rotation <0>: *Enter a value or press* ENTER

**TIP** You can drag a DGN file onto the drawing window to start the DGNATTACH command.

#### DGN File

The file name can include up to 255 characters and contain letters, digits, spaces, and any special characters not used by Microsoft Windows or this program.

Entering a tilde (~) displays the Select DGN File dialog box (a standard file selection dialog box (page 931)).

To avoid errors when entering a file name, it is recommended that you specify both the DGN file and path as follows:

*path name\filename.dgn*

or

*"path name\filename.dgn"*

If you enter a valid DGN file name without a file extension, the program will add the extension and search for the file.

#### ? - List of Design Models

Lists all design models contained in the DGN file.

#### Unit

Specifies a drawing-units value for scaling the DGN underlay.

Enter unit [MM/Centimeter/Meter/Kilometer/Inch/Foot/Yard/MILe/Unitless] <current unit>: *Enter an option or press* ENTER

# DGNCLIP

## Quick Reference

Defines a clipping boundary for a selected DGN underlay

Select an DGN underlay to clip, right-click in the drawing area, and choose DGN Clip.
**dgnclip**
Select DGN to clip: *Use an object selection method and press ENTER* when you finish selecting objects
Enter DGN clipping option [ON (page 392)/OFF (page 392)/Delete (page 392)/New boundary (page 392)] <New boundary>: *Enter an option or press* ENTER

### On

Turns on clipping and displays the DGN underlay clipped to the previously defined boundary.

### Off

Turns off clipping and displays the entire DGN underlay.

If you reclip the DGN underlay while clipping is turned off, clipping is automatically turned back on. You are prompted to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.

### Delete

Removes a predefined clipping boundary and redisplays the entire DGN underlay.

### New Boundary

Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the DGN underlay's boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the point is interpreted as the first corner of a rectangle.

Enter clipping type [Polygonal/Rectangular] <Rectangular>: *Enter* **p** *or press* ENTER

If the DGN underlay already has a clipping boundary defined, the following prompt is displayed:

Delete old boundary? [Yes/No] <Yes>: *Enter* **yes** *or* **no** *and then press* ENTER

If you choose Yes, the entire DGN underlay is redrawn and the command continues; if you choose No, the command ends.

**Polygonal** Uses specified points to define a polygonal boundary.

Specify first point: *Specify a point*

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Close/Undo]: *Specify a point, or enter* **c** *or* **u**

You must specify at least three points to define a polygon.

**Rectangular** Specifies a rectangular boundary by its opposite corners. The rectangle is always drawn parallel to the edges of the DGN underlay.

Specify first corner point: *Specify a point*

Specify opposite corner point: *Specify a point*

# DGNEXPORT

## Quick Reference

Creates one or more V8 DGN files from the current drawing

**dgnexport**

The Export DGN File dialog box (a standard file selection dialog box (page 931)) is displayed. Once you specify a DGN file name, the Export DGN Settings dialog box (page 393) is displayed.

**WARNING** Some programs that work with DGN files do not support extended characters that the Windows operating system considers valid for file names. Thus, it is recommended that you do not use accented or Asian characters in file names when using DGNEXPORT.

# Export DGN Settings Dialog Box

## Quick Reference

**dgnexport**

Controls how objects are processed when exporting them to a V8 DGN file.

### External DWG References

Controls how xrefs are processed.

**Translate All DWG References to DGN Files** All referenced DWG files, including nested DWG references, are converted into DGN files. The nesting relationships of the references are maintained, but all resulting DGN files are created in the same folder. The resulting DGN files use the same file name as the DWG files, but use a *.dgn* file extension.

**NOTE** If a DGN file of the same name already exists in the folder, the DGN file is not overwritten.

**Bind All DWG References into one DGN File** The parent DWG file and all referenced DWG files are combined into a single DGN file. The referenced DWG files will be converted to cells as part of the primary DGN file.

**Ignore DWG References** Referenced DWG files are not included in the resulting DGN file.

### External DGN References

Controls whether DGN underlays in the DWG file are exported as DGN references of the resulting DGN file.

**Export DGN Underlays** When checked, any DGN Underlays are exported as DGN references along with the base drawing. When cleared, DGN underlays are not exported.

### Seed File

A *seed file* for a DGN file is similar to a drawing template (DWT) file that contains default settings and attributes, such as working units. Selecting the appropriate DGN seed file (Imperial, Metric, or user defined), and conversion units is critical to the translation from DWG to DGN. In addition, the seed file contains settings that are not present in a DWG file, and therefore inherited in the exported DGN.

These settings include working units and resolution as well as whether the file is 2D or 3D.

### Conversion Units

Select the appropriate conversion units for the translation. The DGN seed file contains working units (imperial or metric) called *master units* and *sub-units*. One DWG unit is converted to either one master unit or one sub-unit.

**Master Units** Specifies that one DWG drawing unit will convert to one master unit of the specified DGN seed file.

**Sub Units** Specifies that one DWG drawing unit will convert to one sub-unit of the specified DGN seed file.

**NOTE** The master units and the sub-units of the selected DGN seed file are displayed next to the buttons for reference.

- **File name drop-down list.** Specify the name of the DGN seed file. Several sample seed files are provided upon installation.
- **Browse button.** Displays the Select Seed File dialog box (a standard file selection dialog box (page 931)), in which you can select the DGN seed file.

### Sample DGN Seed Files

The following table lists the default DGN seed files and their settings. You may need to edit these settings or obtain the appropriate seed file to suit your conversion requirements.

**Sample DGN Seed Files**

| DGN Seed File Name | Master Unit (label) | Sub Unit (label) | Resolution |
|---|---|---|---|
| *Imperial-Seed2D.dgn* | Feet (') | Inches (") | 304,800 per foot |
| *Imperial-Seed3D.dgn* | Feet (') | Inches (") | 304,800 per foot |
| *Metric-Seed2D.dgn* | Meters (m) | Millimeters (mm) | 1,000 per millimeter |
| *Metric-Seed3D.dgn* | Meters (m) | Millimeters (mm) | 1000 per millimeter |

# DGNEXPORT Conversion Table

## Quick Reference

The following table lists the DGN objects and features that are supported for export, and notes on the scope of the translation.

**DWG to DGN Conversion Table**

| AutoCAD | MicroStation | Notes |
|---|---|---|
| Geometric Objects | Geometric Elements | The following DWG objects are translated into DGN elements: line, xline, ray, polyline, arc, circle, spline (NURBS), ellipse, elliptical arc, point, donut, mline, hatch (including gradient fills), and wipeout. Variable width polylines maintain only their starting width when translated into SmartLines. |
| Layers | Levels | Invalid DWG characters in layer names are converted to spaces. |
| Colors | Colors | Colors are matched as closely as possible. Colors set with the ACI (AutoCAD Color Index) are translated directly to the DGN color index. If TrueColor is used and a direct match of RGB values is available in the DGN color index, the colors are also mapped directly. If a TrueColor match is not available, an RGB value will be added to the DGN color index. |
| Linetypes | Line Styles | Custom linetypes are unsupported and could produce unexpected results. |
| Blocks | Cells | Blocks are exported as shared cells. Dynamic blocks are also exported as cells and lose their dynamic behavior. Invisible, Constant, Verify, and Preset attributes are all converted into tags. |

## DWG to DGN Conversion Table

| AutoCAD | MicroStation | Notes |
|---|---|---|
| Single-line Text Objects, Multiline Text Objects, Text Styles | Text, Text Nodes, Text Styles | The visual integrity of multiline text is maintained. However, if exported multiline text objects are edited in MicroStation, the formatting will be lost. |
| Tables | Tables | Table objects are exported as cell elements composed of lines and text. |
| Fields | Fields | In general, fields are translated as static text. The exceptions are fields that behave identically in both products. This includes the Date fields *CreateDate*, *SaveDate*, and *PlotDate*, and the Document property fields *Author*, *Filename*, and *Filesize*.<br>**NOTE** Use the MicroStation Key-in utility to update text containing fields (field update all). |
| Dimensions, Dimension Styles | Dimensions, Dimension Styles | The size, spacing, style, and shape of dimensions may vary slightly. Dimension associativity is maintained whenever possible, and the correct dimension values are always maintained. |
| Raster Images | Raster Images | These image file types are supported: *.bmp*, *.cal*, *.tif*, *.png*, *.tga*, *.jpg*, *.gif*, *.rlc*, *.bil*, and *.pcx*. All other image file types are discarded. |
| DWG References | DGN References | Depending on the export option specified, all referenced DWG files, including nested DWG references, are either converted into DGN files, combined into a single DGN file as cells, or discarded. |

**DWG to DGN Conversion Table**

| AutoCAD | MicroStation | Notes |
|---|---|---|
| DWG Model | DGN Design Model | The model in a DWG file is converted into a DGN file. |
| DWG Layout | DGN Sheet Model | Any initialized layouts are converted into sheet models in the DGN file. |

### Limitations

The following are general limitations when exporting to V8 DGN drawings:

- Only those objects with Z values of 0 can be translated to 2D seed files.
- 3D solids, including 3D primitives (polysolid, box, wedge, cone, sphere, cylinder, torus, pyramid) and composite 3D solids, are not translated.
- 3D surfaces are not translated.
- 2D and 3D helixes are not translated.
- Translation of Annotative scaling is not supported.
- Translation of Layer property overrides by viewport is not supported.
- Custom AutoCAD objects are not translated. Proxy objects are translated to equivalent DGN objects.

# DGNIMPORT

## Quick Reference

Imports the data from a V8 DGN file into a new DWG file

**dgnimport**

The Import DGN File dialog box (a standard file selection dialog box (page 931)) is displayed. Once you select a DGN file, the Import DGN Settings dialog box (page 399) is displayed.

# Import DGN Settings Dialog Box

## Quick Reference

**dgnimport**

Specifies the import settings for a V8 DGN file.

### Select a Design Model from the DGN file

The DGN design model that you select is imported into the model space of your current DWG file.

DGN files might contain multiple design and sheet models. A DGN design model is equivalent to model space, and a DGN sheet model is equivalent to paper space. Because a DWG file can have only one model space, you need to select the design model in the DGN file that you want to import.

If a design model is selected and is referenced into a sheet model, then the sheet model is also converted as one or more layouts in the DWG file. Only the sheet models that reference the primary design model will be imported. These sheet models will be translated to paper space layouts, and will include layout viewports that display each reference to the primary design model.

### External DGN References

Controls how DGN references are handled in a DWG environment. Support reference file types include *.dgn* and *.dwg* files. All other file types (*.cel*, *.s*, *.h*, *.cgm*, *.rdl*, *.d*, *.dxf*) are not supported and are ignored.

**Ignore External References** Referenced DGN and DWG files that are *external* to the specified DGN file are not included in the resulting DWG file.

If the DGN file has self references or references to other design models contained within the DGN file, these references will still be included.

**Translate References to DWG** All referenced DGN design models, including nested references, are converted into individual DWG files and attached as xrefs in model space.

---

**NOTE** Any DWG files that are referenced by the DGN file are imported as xrefs to those same DWG files.

---

The nesting relationships of these references are maintained. DGN design model references that reference the primary DGN design model, also called

circular references or self references, are handled by creating a separate DWG copy of the DGN design model and attaching it to the primary DWG as an xref. Design model references to sheet models, and sheet model references to other sheet models are not supported and are ignored. Unique names for the referenced DWG files are generated by appending the model name to the design file name separated with a dash (-). If necessary, the resulting DWG file names is truncated and one or more of the last characters might be replaced with numerals to make the file name unique.

**IMPORTANT** Existing DWG files are *not* overwritten when new DWG files are generated by this command. In some cases, this can lead to unexpected results. For example, if you import the same DGN file more than once using different unit settings, the first set of generated DWG files are not overwritten and they will use their original units settings.

**Create DGN Underlay** All DGN external references in the DGN file are imported as DGN underlays in the resulting DWG file. Referenced DWG files remain xrefs when imported.

**NOTE** New DWG files are created in this process. These new DWGs are not automatically deleted.

### Explode Text Node to Text Elements

*Text node elements* in a DGN file are multiple lines of text stored as one element, and are similar to MTEXT objects.

When cleared, multiple lines of text are maintained as a single multiline text object.

When checked, multiple lines of text are separated into individual text objects. Select this option to maintain visual fidelity when you import a DGN file that includes text created along a curved path. The first character determines the location and rotation of the text object.

### Conversion Units

Select the appropriate conversion units for the translation. The imported DGN file contains working units (imperial or metric) called *master units* and *sub-units*. The selected working units (master units or sub-units) are converted one-for-one into DWG units.

**Master Units** Specifies that one master unit of the imported DGN file converts to one DWG drawing unit.

**Sub Units** Specifies that one sub-unit of the imported DGN file converts to one DWG drawing unit.

**NOTE** The master units and the sub-units of the selected DGN file are displayed next to the buttons for reference.

# DGNIMPORT Conversion Table

## Quick Reference

The following table lists the DGN objects and features that are supported for import, and notes on the scope of the translation.

**DGN to DWG Conversion Table**

| MicroStation | AutoCAD | Notes |
|---|---|---|
| Geometric Elements | Geometric Objects | The following DGN elements are translated into DWG objects: line, SmartLine, LineString, Multiline, shape, complex chain, complex shape, arc, ellipse, curve, B-spline curve, and pattern. Pattern elements are limited to simple pattern styles. Bitonal gradient properties are also supported. |
| Levels | Layers | DGN levels are mapped automatically into the equivalent DWG layers. Invalid DGN characters in level names are converted to spaces. |
| Colors | Colors | All colors are translated using RGB values either to the ACI (AutoCAD Color Index) or TrueColor index as appropriate. |
| Line Styles | Linetypes | Only simple line styles using solid lines, dashes, and dots are supported. Unsupported custom line styles are converted into continuous linetypes. |

**DGN to DWG Conversion Table**

| MicroStation | AutoCAD | Notes |
|---|---|---|
| | | Dynamically scaled line styles are converted into fixed-scale linetypes. The resulting linetypes use a scale near the middle of the scaling range of the dynamic line style. |
| Cells | Blocks | Cells are converted into blocks. Tags that are associated with cells are converted into constant-mode block attributes. |
| Text Elements, Text Styles | Single-line Text Objects, Multiline Text Objects, Text Styles | Missing SHX text fonts are converted into the AutoCAD text font specified in the FONTALT system variable. The *acad.fmp* can be used to map SHX text fonts to DGN text fonts. |
| Tables | Tables | Cells that represent tables in DGN are imported as blocks. |
| Dimensions, Dimension Styles | Dimensions, Dimension Styles | The size, spacing, style, and shape of dimensions may vary slightly. Dimension associativity is maintained whenever possible, and the correct dimension values are always maintained. |
| Fields | Fields | In general, fields are translated as static text. The exceptions are fields that are behave identically in both products. This includes the Date fields *CreateDate*, *SaveDate*, and *PlotDate*, and the Document property fields *Author*, *Filename*, and *Filesize*. |
| Raster Images | Raster Images | These image file types: *.bmp*, *.cal*, *.tif*, *.png*, *.tga*, *.jpg*, *.pcx*, *.gif*, *.rlc*, *.bil*, and *.pct* are supported. All other image file types are discarded. |
| DGN References | DWG References | Only *.dwg* and *.dgn* file types are supported. |

**DGN to DWG Conversion Table**

| MicroStation | AutoCAD | Notes |
|---|---|---|
| DGN Design Model | DWG Model | The selected design model in a DGN file is converted into a DWG file. |
| DGN Sheet Model | DWG Layout | Only the sheet models that reference the primary design model are imported. DGN sheet models are translated as layouts in a DWG file. |

### Limitations

The following are general limitations when importing V8 DGN drawings.

- Objects imported as 3D polylines do not display non-continuous linetypes. To see the linetypes, explode the 3D polylines.
- 3D solids, including 3D primitives (slab, sphere, cylinder, cone, torus, wedge) and composite 3D solids, are not translated.
- 3D surfaces are not translated.

# DGNIMPORT Unit Mapping Table

## Quick Reference

The following table lists the linear units used in the resulting DWG file as determined by the master units, sub units, and unit precision of the DGN file being imported.

**DGN to DWG Unit Mapping Table**

| DGN Master Units | DGN Sub Units | DGN Precision | DWG Units | Notes |
|---|---|---|---|---|
| Feet | Inches | Decimal | Engineering | |

**DGN to DWG Unit Mapping Table**

| DGN Master Units | DGN Sub Units | DGN Precision | DWG Units | Notes |
|---|---|---|---|---|
| Feet | Inches | Fractional | Architectural | |
| Inches | Inches | Decimal | Engineering | |
| Inches | Inches | Fractional | Architectural | |
| Inches | Any Other Unit | Decimal | Engineering | Master Units selected in the Import DGN Settings dialog box. |
| Inches | Any Other Unit | Decimal | Decimal | Sub Units selected in the Import DGN Settings dialog box. |
| Inches | Any Other Unit | Fractional | Architectural | Master Units selected in the Import DGN Settings dialog box |
| Inches | Any Other Unit | Fractional | Fractional | Sub Units selected in the Import DGN Settings dialog box |
| Inches | Any Other Unit | Decimal | Decimal | |

**DGN to DWG Unit Mapping Table**

| DGN Master Units | DGN Sub Units | DGN Precision | DWG Units | Notes |
|---|---|---|---|---|
| Any Other Unit | Any Other Unit | Fractional | Fractional | |

# DIM and DIM1

## Quick Reference

Accesses Dimensioning mode

**dim** or **dim1**
Dim: *Enter a Dimensioning mode command*

The Dim prompt indicates that you're in Dimensioning mode, in which you can use a special set of dimensioning commands (page 405). (DIM and DIM1 are provided only for compatibility with previous releases.)

Use DIM to remain in Dimensioning mode after you have used a dimensioning command. Use DIM1 to execute a dimensioning command and immediately return to the Command prompt. To exit Dimensioning mode, enter **e** or **exit**, or press ESC.

### Dimensioning Mode Commands

The following commands are available at the DIM prompt.

**Dimensioning mode commands**

| Command | Description |
|---|---|
| EXIT | Exits Dimensioning mode and returns to the Command prompt. You can also press ESC to exit Dimensioning mode. |
| REDRAW | Redraws the current viewport. |

**Dimensioning mode commands**

| Command | Description |
|---|---|
| STYLE | Changes the current text style. |
| UNDO or U | Erases the most recently created dimension objects and cancels any new dimension system variable setting. When you exit Dimensioning mode, UNDO reverses the effects of the entire dimensioning session. |

The following table shows which AutoCAD commands are equivalent to the rest of the Dimensioning mode commands. For information about these Dimensioning mode commands, see the equivalent AutoCAD command.

**Dimensioning mode command equivalents**

| Dimensioning mode command | Equivalent command |
|---|---|
| ALIGNED | *DIMALIGNED* |
| ANGULAR | *DIMANGULAR* |
| BASELINE | *DIMBASELINE* |
| CENTER | *DIMCENTER* |
| CONTINUE | *DIMCONTINUE* |
| DIAMETER | *DIMDIAMETER* |
| HOMETEXT | *DIMEDIT* Home |
| HORIZONTAL | *DIMLINEAR* Horizontal |
| LEADER | *LEADER* |
| JOG | *DIMJOGGED* |

**Dimensioning mode command equivalents**

| Dimensioning mode command | Equivalent command |
|---|---|
| NEWTEXT | *DIMEDIT* New |
| OBLIQUE | *DIMEDIT* Oblique |
| ORDINATE | *DIMORDINATE* |
| OVERRIDE | *DIMOVERRIDE* |
| RADIUS | *DIMRADIUS* |
| RESTORE | - Restore |
| ROTATED | *DIMLINEAR* Rotated |
| SAVE | -*DIMSTYLE* Save |
| STATUS | -*DIMSTYLE* Status |
| TEDIT | *DIMTEDIT* |
| TROTATE | *DIMEDIT* Rotate |
| UPDATE | -*DIMSTYLE* Apply |
| VARIABLES | -*DIMSTYLE* Variables |
| VERTICAL | *DIMLINEAR* Vertical |

# DIMALIGNED

## Quick Reference

Creates an aligned linear dimension

Dimension 

Dimension ➤ AlignedAt the Command prompt, enter dimaligned.

**dimaligned**

Specify first extension line origin (page 408) or <select object (page 408)>: *Specify a point for manual extension lines, or press* ENTER *for automatic extension lines*

After you specify either manual or automatic extension lines, the following prompt is displayed:

Specify dimension line location (page 409) or [Mtext (page 409)/Text (page 409)/Angle (page 410)]: *Specify a point or enter an option*

### Extension Line Origin

Specifies the first extension line origin (1). You are prompted to specify the second one.

Specify second extension line origin: *Specify a point (2)*

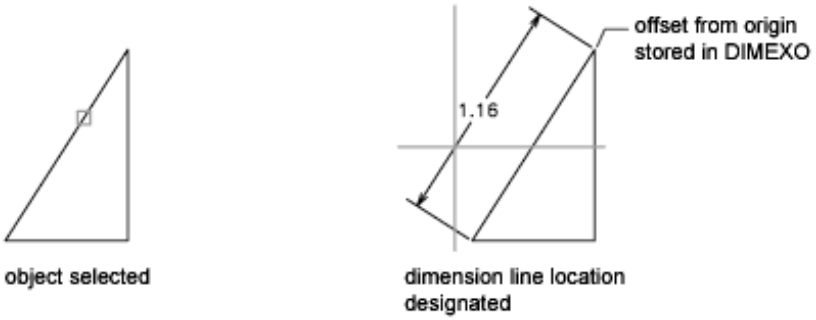
### Object Selection

Automatically determines the origin points of the first and second extension lines after you select an object.

Select object to dimension:

For polylines and other explodable objects, only the individual line and arc segments are dimensioned. You cannot select objects in a nonuniformly scaled block reference.

If you select a line or an arc, its endpoints are used as the origins of the extension lines. The extension lines are offset from the endpoints by the distance specified in Offset from Origin on the Lines and Arrows tab of the New, Modify, and Override Dimension Style dialog boxes (see *DIMSTYLE*). This value is stored in the *DIMEXO* system variable.




If you select a circle, the endpoints of its diameter are used as the origins of the extension line. The point used to select the circle defines the origin of the first extension line.

### Dimension Line Location

Specifies the placement of the dimension line and determines the direction to draw the extension lines. After you specify the location, the DIMALIGNED command ends.

### Mtext

Displays the In-Place Text Editor (page 873), which you can use to edit the dimension text. The generated measurement is represented with angle brackets (< >). To add a prefix or a suffix to the generated measurement, enter the prefix or suffix before or after the angle brackets. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the angle brackets, enter the new dimension text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more information about formatting dimension text, see “Change Existing Objects” in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

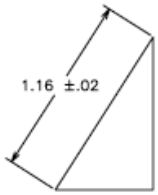
Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.


before angle

After you specify the angle, the Dimension Line Location prompt is redisplayed.




after angle

# DIMANGULAR

## Quick Reference

Creates an angular dimension

Dimension 

Dimension ➤ AngularAt the Command prompt, enter dimangular.

**dimangular**

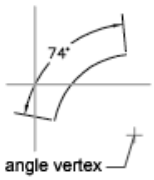
Select arc (page 411), circle (page 411), line (page 412), or <specify vertex (page 412)>: *Select an arc, circle, or line, or press* ENTER *to create the angular dimension by specifying three points*

After you define the angle to dimension, the following prompt is displayed:

Specify dimension arc line location (page 412) or [Mtext (page 413)/Text (page 413)/Angle (page 413)/Quadrant (page 414)]:

### Arc Selection

Uses points on the selected arc as the defining points for a three-point angular dimension. The center of the arc is the angle vertex. The arc endpoints become the origin points of the extension lines.



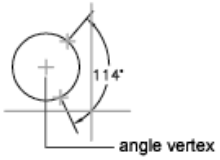

The dimension line is drawn as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

### Circle Selection

Uses the selection point (1) as the origin of the first extension line. The center of the circle is the angle vertex.

Specify second angle endpoint: *Specify a point (2)*

The second angle endpoint is the origin of the second extension line and does not have to lie on the circle.

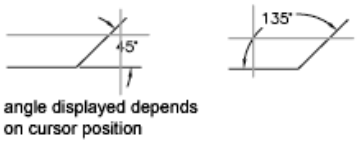


### Line Selection

Defines the angle using two lines.

Select second line:

The program determines the angle by using each line as a vector for the angle and the intersection of the lines for the angle vertex. The dimension line spans the angle between the two lines. If the dimension line does not intersect the lines being dimensioned, extension lines are added as needed to extend one or both lines. The arc is always less than 180 degrees.

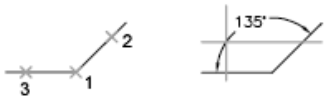



### Three-Point Specification

Creates a dimension based on three points you specify.

Specify angle vertex: *Specify a point (1)*
Specify first angle endpoint: *Specify a point (2)*
Specify second angle endpoint: *Specify a point (3)*

The angle vertex can be the same as one of the angle endpoints. If you need extension lines, the endpoints are used as origin points.




The dimension line is drawn as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

### Dimension Arc Line Location

Specifies the placement of the dimension line and determines the direction to draw the extension lines. After you specify the location, DIMANGULAR ends.

### Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. For more information about formatting dimension text, see "Change Existing Objects" in the *User's Guide*.

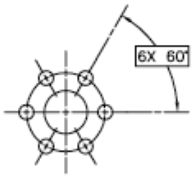
The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement.



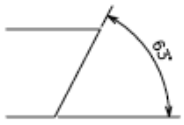

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

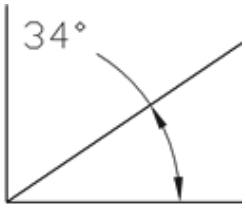
Enter an angle. For example, to rotate the text 45 degrees, enter **45**.



**Quadrant**

Specifies the quadrant that the dimension should be locked to. When quadrant behavior is on, the dimension line is extended past the extension line when the dimension text is positioned outside of the angular dimension.

Specify quadrant: *Specify a quadrant*




# DIMARC

## Quick Reference

Creates an arc length dimension

Dimension 

Dimension ➤ Arc LengthAt the Command prompt, enter dimarc.

**dimarc**

Select arc or polyline arc segment: *Use an object selection method*

Specify arc length dimension location (page 414) or [Mtext (page 415)/Text (page 415)/Angle (page 415)/Partial (page 415)/Leader (page 416)]: Specify a point or enter an option

**Arc Length Dimension Location**

Specifies the placement of the dimension line and determines the direction of the extension lines.

### Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([]). For more information about formatting dimension text, see "Change Existing Objects" in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

### Partial

Reduces the length of the arc length dimension.

Specify first point for arc length dimension:

Specify a point on the arc where the arc length dimension should begin.

Specify second point for arc length dimension:

Specify a point on the arc where the arc length dimension should end.

**Leader**

Adds a leader object. This option is displayed only if the arc (or arc segment) is greater than 90 degrees. The leader is drawn radially, pointing towards the center of the arc being dimensioned.

Specify arc length dimension location or [Mtext/Text/Angle/Partial/No leader]:

Specify a point or enter an option. The leader is created automatically. The No Leader option cancels the Leader option before the leader is created. To remove a leader, delete the arc length dimension and recreate it without the leader option.

# DIMBASELINE

## Quick Reference

Creates a linear, angular, or ordinate dimension from the baseline of the previous dimension or a selected dimension

Dimension 
Dimension ➤ BaselineAt the Command prompt, enter dimbaseline.
**dimbaseline**

If no dimension was created in the current session, you are prompted to select a linear, ordinate, or angular dimension to use as the base for the baseline dimension.

Select base dimension: *Select a linear, ordinate, or angular dimension*

Otherwise, the program skips this prompt and uses the dimension object that was last created in the current session. If the base dimension is linear or angular, the following prompt is displayed:

Specify a second extension line origin (page 417) or [Undo (page 418)/Select (page 418)] <Select>: *Specify a point, enter an option, or press* ENTER *to select a base dimension*

If the base dimension is ordinate, the following prompt is displayed:

Specify feature location (page 417) or [Undo/Select] <Select>:

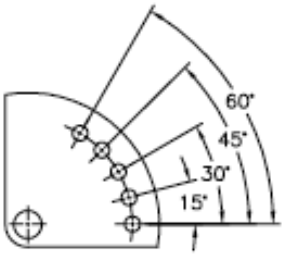
To end the command, press ENTER twice, or press ESC. The current dimension style determines the appearance of the text.

### Second Extension Line Origin

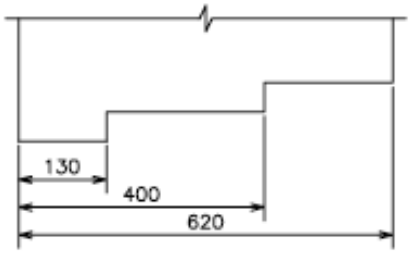
By default, the first extension line of the base dimension is used as the extension line origin for the baseline dimension. To override this default behavior, explicitly select the base dimension; the extension line origin becomes the extension line of the base dimension closest to the pick point of the selection. When you select a second point, the baseline dimension is drawn and the Specify a Second Extension Line Origin prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the baseline dimension, press ENTER.

Select base dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.




angular baseline dimensioning




linear baseline dimensioning

### Feature Location

Uses the endpoint of the base dimension as the endpoint for the baseline dimension; you are prompted for the next feature location. When you select a feature location, the baseline dimension is drawn and the Specify Feature Location prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the baseline dimension, press ENTER.

Select base dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.

#### Undo

Undoes the last baseline dimension entered during this command session.

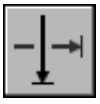
#### Select

Prompts you to select a linear, ordinate, or angular dimension to use as the base for the baseline dimension. After you select a base dimension, the Specify Second Extension Line Origin prompt or the Specify Feature Location prompt is redisplayed.

# DIMBREAK

## Quick Reference

Adds or removes a dimension break

Dimension 

Dimension ➤ Dimension BreakAt the Command prompt, enter DIMBREAK.
**dimbreak**
Select a dimension or [Multiple (page 418)]: *Select a dimension, or enter* **m** *and press* ENTER

After you select a dimension, the following prompt is displayed:

Select object to break dimension or [Auto (page 419)/Restore (page 419)/Manual (page 419)] <Auto>: *Select an object that intersects the dimension or extension lines of the selected dimension, enter an option, or press* ENTER

After you select an object to break the dimension with, the following prompt is displayed:

Select object to break dimension: *Select an object that passes through the dimension or press* ENTER *to end the command*

---

**NOTE** Dimension breaks can be added to dimensions for objects that do not intersect the dimension or extension lines using the by Manual option.

---

#### Multiple

Specifies multiple dimensions to add breaks to or remove breaks from.

Select dimensions: *Use an object selection method and press* ENTER
Enter an option [Break (page 419)/Restore (page 419)] <Break>: *Enter an option or press* ENTER

### Break

Places dimension breaks automatically at all the intersection points of the objects that intersect the selected dimension. Any dimension breaks that are created using this option are updated automatically when the dimension or intersecting object is modified.

When a new object is drawn over the top of a dimension that has any dimension breaks, no new dimension breaks are automatically applied at the intersecting points along the dimension object. To add new dimension breaks, must be run the command again.

### Auto

Places dimension breaks automatically at all the intersection points of the objects that intersect the selected dimension. Any dimension break created using this option is updated automatically when the dimension or an intersecting object is modified.

When a new object is drawn overthe top of a dimension that has any dimension breaks, no new dimension breaks are automatically applied at the intersecting points along the dimension object. To add the new dimension breaks, must be run the command again.

### Restore

Removes all dimension breaks from the selected dimensions.

### Manual

Places a dimension break manually. You specify two points on the dimension or extension lines for the location of the break. Any dimension break that is created using this option is not updated if the dimension or intersecting objects are modified. You can only place a single manual dimension break at a time with this option.

Specify first break point: *Specify a point*
Specify second break point: *Specify a point*

# DIMCENTER

## Quick Reference

Creates the center mark or the centerlines of circles and arcs

Dimension 

Dimension ➤ Center MarkAt the Command prompt, enter dimcenter.

**dimcenter**

Select arc or circle: *Use an object selection method*

You can choose between center marks and centerlines and specify their size when you set up the dimension style. See *DIMSTYLE*. You can also change center mark settings using the *DIMCEN* system variable.

center mark

centerlines

# DIMCONTINUE

## Quick Reference

Creates a linear, angular, or ordinate dimension from the second extension line of the previous dimension or a selected dimension

Dimension 

Dimension ➤ ContinueAt the Command prompt, enter dimcontinue.

**dimcontinue**

If no dimension was created in the current session, you are prompted to select a linear, ordinate, or angular dimension to use as the base for the continued dimension.

Select continued dimension: *Select a linear, ordinate, or angular dimension*

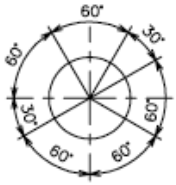
Otherwise, the program skips this prompt and uses the dimension object that was last created in the current session. If the base dimension is linear or angular, the following prompt is displayed:

Specify a second extension line origin (page 421) or [Undo (page 422)/Select (page 422)] <Select>: *Specify a point, enter an option, or press* ENTER *to select a base dimension*

If the base dimension is ordinate, the following prompt is displayed:

Specify feature location (page 421) or [Undo/Select] <Select>:

To end the command, press ENTER twice, or press ESC. The current dimension style determines the appearance of the text.



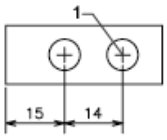

### Second Extension Line Origin

Uses the second extension line origin of the continued dimension for the first extension line origin of the next dimension. The current dimension style determines the appearance of the text.

After you select a continued dimension, the Specify a Second Extension Line Origin prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the continued dimension, press ENTER.

Select continued dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.




### Feature Location

Uses the endpoint of the base dimension as the endpoint for the continued dimension; you are prompted for the next feature location. When you select a feature location, the continued dimension is drawn and the Specify Feature

Location prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the continued dimension, press ENTER.

Select continued dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.

#### Undo

Undoes the last continued dimension entered during the command session.

#### Select

Prompts you to select a linear, ordinate, or angular dimension to use as the continued dimension. After you select a continued dimension, the Specify a Second Extension Line Origin prompt or the Specify Feature Location prompt is redisplayed. To end the command, press ESC.

# DIMDIAMETER

## Quick Reference

Creates diameter dimensions for circles and arcs

Dimension 

Dimension ➤ DiameterAt the Command prompt, enter dimdiameter.

**dimdiameter**

Select arc or circle:

DIMDIAMETER measures the diameter of the selected circle or arc and displays the dimension text with a diameter symbol in front of it.

Specify dimension line location (page 422) or [Mtext (page 423)/Text (page 423)/Angle (page 423)]: *Specify a point or enter an option*

#### Dimension Line Location

Determines the angle of the dimension line and the location of the dimension text. If the dimension is placed off of an arc resulting in the dimension pointing outside the arc, AutoCAD automatically draws an arc extension line.

### Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([]). For more information about formatting dimension text, see "Change Existing Objects" in the *User's Guide*.

The current dimension style determines the appearance of the dimension text.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

After you specify the angle, the Dimension Line Location prompt is redisplayed.

# DIMDISASSOCIATE

## Quick Reference

Removes associativity from selected dimensions

**dimdisassociate**
Select dimensions to disassociate: *Select one or more dimensions and then press* ENTER

DIMDISASSOCIATE filters the selection set to include only associative dimensions that are not on locked layers, and that are not in a different space than the current space (for example, if model space is active, associative dimensions in paper space are excluded). DIMDISASSOCIATE then disassociates these dimensions and reports the number of dimensions that are filtered out and the number that are disassociated.

# DIMEDIT

## Quick Reference

Edits dimension text and extension lines on dimension objects

Dimension 
**dimedit**
Enter type of dimension editing [Home (page 424)/New (page 425)/Rotate (page 425)/Oblique (page 425)] <Home>: *Enter an option or press* ENTER
**Home** Moves rotated dimension text back to its default position.

Select objects: *Use an object selection method to select dimension objects*

The selected dimension text is returned to its default position and rotation as specified in its dimension style.

**New** Changes dimension text using the In-Place Text Editor (page 873).


The generated measurement is represented with angle brackets (< >). To add a prefix or a suffix to the generated measurement, enter the prefix or suffix before or after the angle brackets. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

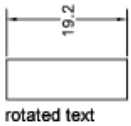
To edit or replace the generated measurement, delete the angle brackets, enter the new dimension text, and then choose OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more information about formatting dimension text, see Change Existing Objects in the *User's Guide*.

**Rotate** Rotates dimension text. This option is similar to the Angle option of *DIMTEDIT*.

Enter text angle:

Entering **0** places the text in its default orientation, which is determined by the vertical and horizontal text settings on the Text tab of the New, Modify, and Override Dimension Style dialog boxes. See *DIMSTYLE*. The *DIMTIH* and *DIMTOH* system variables control this orientation.

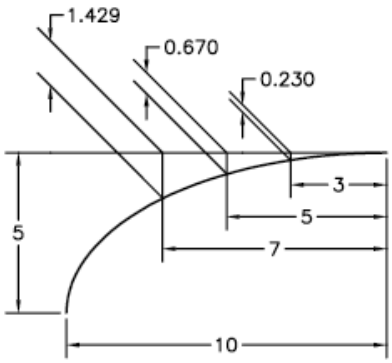
Select objects: *Use an object selection method to select dimension objects*




**Oblique** Adjusts the oblique angle of the extension lines for linear dimensions. Linear dimensions are created with extension lines perpendicular to the direction of the dimension line. The Oblique option is useful when extension lines conflict with other features of the drawing.

Select objects: *Use an object selection method to select dimension objects*

Enter oblique angle (press ENTER for none): *Enter an angle or press* ENTER




# DIMINSPECT

## Quick Reference

Create or remove inspection dimensions

Dimension 

Dimension ➤ InspectionAt the Command prompt, enter diminspect.
**diminspect**

The Inspection Dimension Dialog Box (page 427) is displayed and allows youi to add or remove inspection dimensions from an existing dimension. Inspection dimensions allow you to effectively communicate how frequently manufactured parts should be checked to ensure that the dimension value and tolerances of the parts are within the specified range.

If you enter **-diminspect** at the command prompt, options are displayed at the command prompt (page 428).

# Inspection Dimension Dialog Box

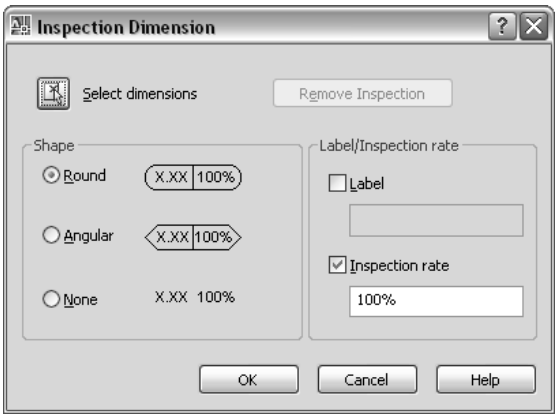
## Quick Reference

Dimension

Dimension ➤ InspectionAt the Command prompt, enter diminspect.

**diminspect**

Allows you to add or remove an inspection dimension from selected dimensions. Use the Shape and Inspection Label/Rate settings to the appearance of the frame of the inspection dimension and the inspection rate value.




### Select Dimensions

Specifies the dimensions that an inspection dimension should be added to or removed from.

You can select dimensions before or after the Inspection Dimension dialog box is displayed. To select dimensions when the Inspection Dimension dialog box is displayed, click Select Dimensions. The dialog box temporarily closes. When you finish selecting dimensions, press ENTER. The Inspection Dimension dialog box is redisplayed.

#### Remove Inspection

Removes the inspection dimension from the selected dimension.

#### Shape

Controls the shape of the frame that is drawn around the label, dimension value, and inspection rate of the inspection dimension.

**Round** Creates a frame with semi-circles on the two ends; the fields within the frame are separated by vertical lines.

**Angular** Creates a frame with lines that form a 90-degree angle on the two ends; the fields within the frame are separated by vertical lines.

**None** Specifies that no frame is drawn around the values; the fields are not separated by vertical lines.

#### Label/Inspection Rate

Specifies the label text and inspection rate for an inspection dimension.

**Label** Turns the display of the label field on and off.

**Label Value** Specifies the label text.

The label is displayed in the leftmost section of the inspection dimension when the Label check box is selected.

**Inspection Rate** Turns the display of the rate field on and off.

**Inspection Rate Value** Specifies how frequent a part should be inspected.

The value is expressed as a percentage, and the valid range is 0 to 100. The inspection rate is displayed in the rightmost section of the inspection dimension when the Inspection Rate check box is selected.

## -DIMINSPECT

### Quick Reference

If you enter **-diminspect** at the command prompt, the following DIMINSPECT command prompts are displayed.

Add inspection data (page 429) or [Remove (page 429)] <Add>: *Enter an option or press* ENTER

### Add

Adds an inspection dimension to the selected dimensions.

Select dimensions: *Use an object selection method to select dimension objects and press* ENTER *when you finish*

After you select dimension objects, the following prompt is displayed:

Enter shape option [Round/Angular/None] <Round>: *Enter an option or press* ENTER
**Round** Creates a frame with semi-circles on the two ends; the fields within the frame are separated by vertical lines.

**Angular** Creates a frame with lines that form a 90-degree angle on the two ends; the fields within the frame are separated by vertical lines.

**None** Specifies that no frame is drawn around the values; the fields are not separated by vertical lines.

After you specify a frame shape, the following prompt is displayed:

Enter label data or <None>: *Enter a value for the inspection label or press* ENTER
**None** No label is displayed with the inspection dimension.

After you specify a label, the following prompt is displayed:

Enter inspection rate <100%>: *Enter a value for the inspection rate or press* ENTER

### Remove

Removes the inspection dimension from the selected dimensions.

Select dimensions: *Use an object selection method and press ENTER when you finish*

After you press ENTER, a status string of the number of inspection dimensions that were removed is displayed at the command prompt.

*n* of inspection removed

*n* represents the number of inspections removed from the selected dimension objects.

# DIMJOGGED

## Quick Reference

Creates jogged radius dimensions

Dimension 

Dimension ➤ JoggedAt the Command prompt, enter dimjogged.

**dimjogged**

Select arc or circle: Select an arc, circle, or polyline arc segment

DIMJOGGED measures the radius of the selected object and displays the dimension text with a radius symbol in front of it. The origin point of the dimension line can be specified at any convenient location.

**NOTE** Jogged radius dimensions are also called *foreshortened radius dimensions*.

Specify center location override: *Specify a point*

Accepts a new center point for a jogged radius dimension that takes the place of the true center point of the arc or circle.

Specify dimension line location (page 430) or [Mtext (page 430)/Text (page 431)/Angle (page 431)]: *Specify a point or enter an option*

### Dimension Line Location

Determines the angle of the dimension line and the location of the dimension text. If the dimension is placed off of an arc resulting in the dimension pointing outside the arc, AutoCAD automatically draws an arc extension line.

### Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more

information about formatting dimension text, see “Change Existing Objects” in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>: *Enter the dimension text, or press* ENTER *to accept the generated measurement*

To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text: *Enter an angle*

For example, to rotate the text 45 degrees, enter **45**.

Also determines the angle of the dimension line and the location of the dimension text.

Specify jog location: *Specify a point*

Locates the middle point of the jog. The transverse angle of the jog is determined by the Dimension Style Manager.

# DIMJOGLINE

## Quick Reference

Adds or removes a jog line on a linear or aligned dimension

Dimension 
Dimension ➤ Jogged LinearAt the Command prompt, enter DIMJOGLINE.
**dimjogline**
Select dimension to add jog (page 432) or [Remove (page 432)]: *Select a linear or aligned dimension*

#### Add Jog

Specifies the linear or aligned dimension to which to add a jog. You are prompted for the location of the jog.

Specify jog location (or press ENTER): *Specify a point for the location of the jog, or press ENTER to place the jog at the midpoint between the dimension text and the first extension line or the midpoint of the dimension line based on the location of the dimension text*

#### Remove

Specifies the linear or aligned dimension from which to remove the jog.

Select jog to remove: *Select a linear or aligned dimension*

# DIMLINEAR

## Quick Reference

Creates a linear dimensions

Dimension 
Dimension ➤ LinearAt the Command prompt, enter dimlinear.
**dimlinear**
Specify first extension line origin (page 433) or <select object (page 436)>: *Specify a point or press* ENTER *to select an object to dimension*

After you specify the extension line origin points or the object to dimension, the following prompt is displayed:

Specify dimension line location (page 433) or [Mtext (page 433)/Text (page 434)/Angle (page 434)/Horizontal (page 434)/Vertical (page 435)/Rotated (page 435)]: *Specify a point or enter an option*

### First Extension Line Origin

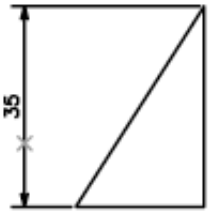
Prompts for the origin point of the second extension line after you specify the origin point of the first.

Specify second extension line origin: *Specify a point (2)*

point specification

### Dimension Line Location

Uses the point you specify to locate the dimension line and determines the direction to draw the extension lines. After you specify the location, the dimension is drawn.



dimension line location designated

### Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more information about formatting dimension text, see Change Existing Objects in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

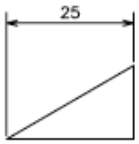
Enter an angle. For example, to rotate the text 90 degrees, enter **90**.


before Angle after Angle

### Horizontal

Creates horizontal linear dimensions.

Specify dimension line location or [Mtext/Text/Angle]: *Specify a point or enter an option*



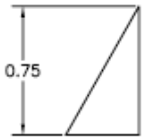

**Dimension Line Location** Uses the point you specify to locate the dimension line. After you specify the location, the dimension is drawn.

**Mtext, Text, Angle** These text editing and formatting options are identical in all dimension commands. See the option descriptions provided earlier in this command.

### Vertical

Creates vertical linear dimensions.

Specify dimension line location or [Mtext/Text/Angle]: *Specify a point or enter an option*



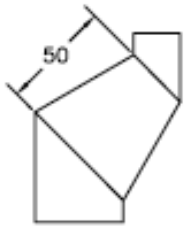

**Dimension Line Location** Uses the point you specify to locate the dimension line. After you specify the location, the dimension is drawn.

**Mtext, Text, Angle** These text editing and formatting options are identical in all dimension commands. See the option descriptions provided earlier in this command.

### Rotated

Creates rotated linear dimensions.

Specify angle of dimension line <*current*>: *Specify an angle or press* ENTER



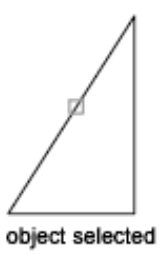

**Object Selection**

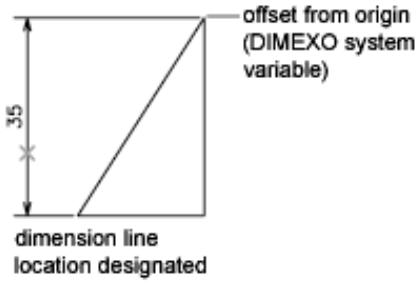
Automatically determines the origin points of the first and second extension lines after you select an object.

Select object to dimension:

For polylines and other explodable objects, only the individual line and arc segments are dimensioned. You cannot select objects in a non-uniformly scaled block reference.

If you select a line or an arc, the line or arc endpoints are used as the origins of the extension lines. The extension lines are offset from the endpoints by the distance you specify in Offset from Origin in the Lines and Arrows tab of the New, Modify, and Override Dimension Style dialog boxes. See *DIMSTYLE*. This value is stored in the *DIMEXO* system variable.



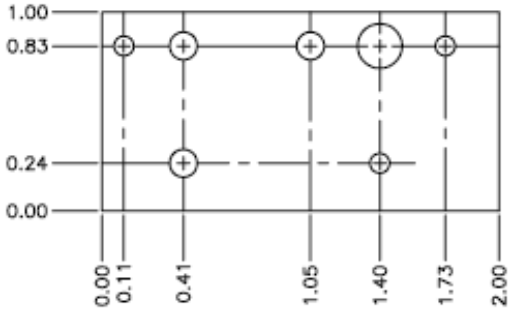




If you select a circle, the diameter endpoints are used as the origins of the extension line. When the point used to select the circle is close to the north or south quadrant point, a horizontal dimension is drawn. When the point used to select the circle is close to the east or west quadrant point, a vertical dimension is drawn.

# DIMORDINATE

## Quick Reference

Creates ordinate point dimensions




Dimension 


Dimension ➤ OrdinateAt the Command prompt, enter dimordinate.

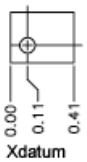
**dimordinate**

Specify feature location: *Specify a point or snap to an object*

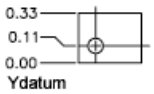
Specify leader endpoint (page 437) or [Xdatum (page 437)/Ydatum (page 438)/Mtext (page 438)/Text (page 438)/Angle (page 438)]: *Specify a point or enter an option*

**Leader Endpoint Specification** Uses the difference between the feature location and the leader endpoint to determine whether it is an *X* or a *Y* ordinate dimension. If the difference in the *Y* ordinate is greater, the dimension measures the *X* ordinate. Otherwise, it measures the *Y* ordinate.

**Xdatum** Measures the *X* ordinate and determines the orientation of the leader line and dimension text. The Leader Endpoint prompt is displayed, where you can specify the endpoint.



**Ydatum** Measures the *Y* ordinate and determines the orientation of the leader line and dimension text. The Leader Endpoint prompts are displayed, where you can specify the endpoint.




**Mtext** Displays the In-Place Text Editor (page 873), which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more information about formatting dimension text, see Change Existing Objects in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements. After you choose OK, the Leader Endpoint prompt is redisplayed.

**Text** Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>: *Enter the dimension text, or press* ENTER to accept the generated measurement

To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes. After you press ENTER, the Leader Endpoint prompt is redisplayed.

**Angle** Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

After you specify the angle, the Leader Endpoint prompt is redisplayed.

# DIMOVERRIDE

## Quick Reference

Overrides dimensioning system variables

Dimension ➤ OverrideAt the Command prompt, enter dimoverride.
**dimoverride**
Enter dimension variable name to override (page 439) or [Clear overrides (page 439)]: *Enter the name of a dimension variable, or enter* **c**
**Dimension Variable Name to Override** Overrides the value of the dimensioning system variable you specify.

Enter new value for dimension variable <*current*>: *Enter a value or press* ENTER

If you enter a new value, the Dimension Variable Name to Override prompt is redisplayed. If you press ENTER, you are prompted to select the dimensions.

Select objects: *Use an object selection method to select the dimensions*

The overrides to the selected dimensions are applied.

**Clear Overrides** Clears any overrides on selected dimensions.

Select objects: *Use an object selection method to select the dimensions*

The overrides are cleared, and the dimension objects return to the settings defined by their dimension style.

# DIMRADIUS

## Quick Reference

Creates radial dimensions for circles and arcs

Dimension 
Dimension ➤ RadiusAt the Command prompt, enter dimradius.
**dimradius**
Select arc or circle:

DIMRADIUS measures the radius of the selected arc or circle and displays the dimension text with a radius symbol in front of it.

Specify dimension line location (page 440) or [Mtext (page 440)/Text (page 440)/Angle (page 440)]: *Specify a point or enter an option*

### Dimension Line Location

Determines the angle of the dimension line and the location of the dimension text. If the dimension is placed off of an arc resulting in the dimension pointing outside the arc, AutoCAD automatically draws an arc extension line.

### Mtext

Displays the In-Place Text Editor (page 873), which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters (page 1431).

To edit or replace the generated measurement, delete text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]). For more information about formatting dimension text, see "Change Existing Objects" in the *User's Guide*.

The current dimension style determines the appearance of the generated measurements.

### Text

Customizes the dimension text at the command prompt. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>: *Enter the dimension text, or press* ENTER *to accept the generated measurement*

To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

### Angle

Changes the angle of the dimension text.

Specify angle of dimension text: *Enter an angle*

For example, to rotate the text 45 degrees, enter **45**.

# DIMREASSOCIATE

## Quick Reference

Associates selected dimensions to geometric objects

Dimension ➤ Reassociate DimensionsAt the Command prompt, enter dimreassociate.
**dimreassociate**
Select dimensions to reassociate: *Select dimension objects*

Each selected dimension is highlighted in turn, and prompts for association points appropriate for the selected dimension are displayed. A marker is displayed for each association point prompt. If the definition point of the current dimension is not associated to a geometric object, the marker appears as an X; if the definition point is associated, the marker appears as an X inside a box.

**NOTE** The marker disappears if you pan or zoom with a wheel mouse.

Press ESC to terminate the command without losing the changes that were already specified. Use *UNDO* to restore the previous state of the changed dimensions.

The prompts for the different types of dimensions are:

**Linear** Specify first extension line origin or [Select object] <next>: *Specify an object snap location, enter* **s** *and select a geometric object, or press* ENTER *to skip to the next prompt*

Specify second extension line origin <next>: *Specify an object snap location, or press* ENTER *to skip to the next dimension object, if any*

**Aligned** Specify first extension line origin or [Select object] <next>: *Specify an object snap location, enter* **s** *and select a geometric object, or press* ENTER *to skip to the next prompt*

Specify second extension line origin <next>: *Specify an object snap location, or press* ENTER *to skip to the next dimension object, if any*

**Angular (Three Point)** Specify angle vertex or [Select arc or circle] <next>: *Specify an object snap location, enter* **s** *and select an arc or a circle, or press* ENTER *to skip to the next prompt*

Specify first angle endpoint <next>: *Specify an object snap location or press* ENTER *to skip to the next prompt*

Specify second angle endpoint <next>: *Specify an object snap location or press* ENTER *to skip to the next dimension object, if any*

**Angular (Two Line)** Select first line <next>: *Select a line, or press* ENTER *to skip to the next prompt*

Select second line <next>: *Select another line, or press* ENTER *to skip to the next dimension object, if any*

**Diameter** Select arc or circle <next>: *Select an arc or a circle, or press* ENTER *to skip to the next dimension object, if any*

**Leader** Specify leader association point <next>: *Specify an object snap location, or press* ENTER *to skip to the next dimension object, if any*

**Ordinate** Specify feature location <next>: *Specify an object snap location, or press* ENTER *to skip to the next dimension object, if any*

**Radius** Select arc or circle <next>: *Select an arc or a circle, or press* ENTER *to skip to the next dimension object, if any*

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**NOTE** DIMREASSOCIATE does not change the setting of *DIMLFAC* in a dimension. Use *DIMOVERRIDE* to clear dimension linear factors in legacy drawings.

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# DIMREGEN

## Quick Reference

Updates the locations of all associative dimensions

**dimregen**

The locations of all associative dimensions in the current drawing are updated.

Associative dimensions need to be updated manually with DIMREGEN in the following three cases:

- After panning or zooming with a wheel mouse in a layout with model space active; update associative dimensions created in paper space.

- After opening a drawing that has been modified with a previous version of the program; update associative dimensions if the dimensioned objects have been modified.
- After opening a drawing containing external references that are dimensioned in the current drawing; update associative dimensions if the associated external reference geometry has been modified.

# DIMSPACE

## Quick Reference

Adjusts the spacing equally between parallel linear and angular dimensions

Dimension 

Dimension ➤ Dimension SpaceAt the Command prompt, enter DIMSPACE.

**dimspace**

Select base dimension: *Select a parallel linear or angular dimension*

Select dimensions to space: *Select a parallel linear or angular dimension to equally space from the base dimension and press* ENTER

Enter value (page 443) or [Auto (page 443)] <Auto>: *Specify a spacing distance or press* ENTER

### Enter Spacing Value

Specifies a spacing value to equally space the selected dimensions from the base dimension. For example, if you enter a value of 0.5000, all selected dimensions will be seperated by a distance of 0.5000.

**NOTE** You can use a spacing value of 0 (zero) to align selected linear and angular dimensions end to end.

### Auto

Calculates the spacing distance automatically based on the text height specified in the dimension style of the selected base dimension. The resulting spacing value is twice the height of the dimension text.

# DIMSTYLE

### Quick Reference

Creates and modifies dimension styles

Styles 
Format ➤ Dimension StyleAt the Command prompt, enter dimstyle.
Dimension ➤ Dimension StyleAt the Command prompt, enter dimstyle.
**dimstyle**

The Dimension Style Manager (page 444) is displayed.

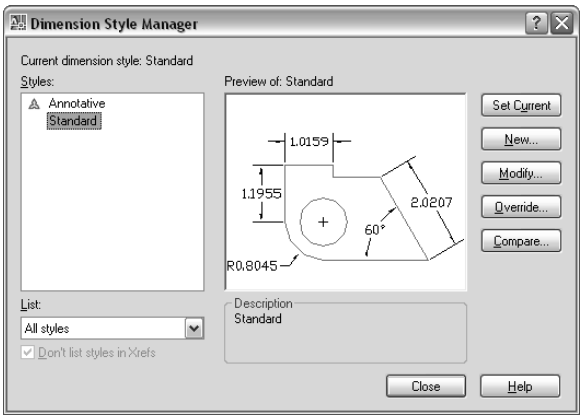
If you enter **-dimstyle** at the command prompt, options are displayed at the command prompt (page 473).

## Dimension Style Manager

### Quick Reference

Styles 
Format ➤ Dimension StyleAt the Command prompt, enter dimstyle.
Dimension ➤ Dimension StyleAt the Command prompt, enter dimstyle.
**dimstyle**

Creates new styles, sets the current style, modifies styles, sets overrides on the current style, and compares styles.




### Current Dimension Style

Displays the name of the dimension style that is current. The default dimension style is STANDARD. The current style is applied to dimensions you create.

### Styles

Lists dimension styles in the drawing. The current style is highlighted. Right-click in the list to display a shortcut menu with options to set the current style, rename styles, and delete styles. You cannot delete a style that is current or in use in the current drawing. A  icon before the style name indicates that the style is .

The item selected in List controls the dimension styles displayed. To make a style current, select it and click Set Current.

Unless you select Don't List Styles in Xrefs, dimension styles are displayed in externally referenced drawings using the syntax for externally referenced named objects. (See “Overview of Referenced Drawings (Xrefs)” in the *User's Guide*.) Although you cannot change, rename, or make current externally referenced dimension styles, you can create new styles based on them.

### List

Controls the display of styles in the Styles list. Select All Styles if you want to see all dimension styles in a drawing. Select Styles in Use if you want to see only the dimension styles currently used by dimensions in the drawing.

### Don't List Styles in Xrefs

When selected, suppresses display of dimension styles in externally referenced drawings in the Styles list.

### Preview

Shows a graphic representation of the style selected in the Styles list.

### Description

Describes the style selected in the Styles list relative to the current style. If the description is longer than the space provided, you can click in the pane and use arrow keys to scroll down.

### Set Current

Sets the style selected under Styles to current. The current style is applied to dimensions you create.

### New

Displays the Create New Dimension Style dialog box (page 447), in which you can define a new dimension style.

### Modify

Displays the Modify Dimension Styles dialog box (page 448), in which you can modify dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box.

### Override

Displays the Override Current Style dialog box (page 448), in which you can set temporary overrides to dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box. Overrides are displayed as unsaved changes under the dimension style in the Styles list.

### Compare

Displays the Compare Dimension Styles dialog box (page 471), in which you can compare two dimension styles or list all the properties of one dimension style.

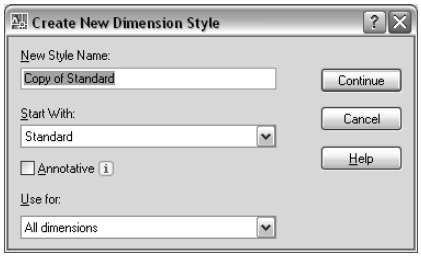
# Create New Dimension Style Dialog Box

## Quick Reference

Styles 

Format ➤ Dimension StyleAt the Command prompt, enter dimstyle.
Dimension ➤ Dimension StyleAt the Command prompt, enter dimstyle.
**dimstyle**

Names the new dimension style, sets the style on which to start the new one, and indicates the dimension types to which you want the new style to apply.




**New Style Name** Specifies the new dimension style name.

**Start With** Sets a style to use as a basis for the new one. For the new style, you change only the properties that differ from the properties you start with.

**Annotative** Specifies that the dimension style is . Click the information icon to learn more about annotative objects.

**Use For** Creates a dimension substyle that applies only to specific dimension types. For example, you could create a version of the STANDARD dimension style to be used only for diameter dimensions.

**Continue** Displays the New Dimension Style dialog box (page 448), in which you define the new dimension style properties.

# New, Modify, and Override Dimension Style Dialog Boxes

## Quick Reference

Styles 
Format ➤ Dimension StyleAt the Command prompt, enter dimstyle.
Dimension ➤ Dimension StyleAt the Command prompt, enter dimstyle.
**dimstyle**

Set properties for dimension styles. When you choose Continue in the Create New Dimension Style dialog box, the New Dimension Style dialog box is displayed. You define the properties for the new style in this dialog box. The dialog box initially displays the properties of the dimension style that you selected to start the new style in the Create New Dimension Style dialog box.

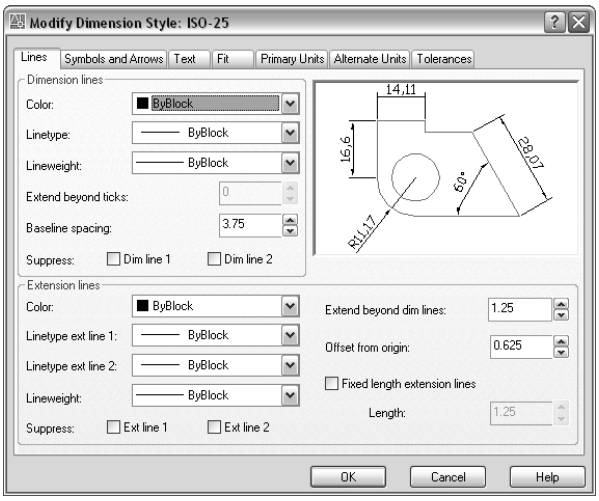
Choosing either Modify or Override in the Dimension Style Manager displays the Modify Dimension Style or the Override Dimension Style dialog box. The content of these dialog boxes is identical to the New Dimension Style dialog box, although you are modifying or overriding an existing dimension style rather than creating a new one.

The sample image on each tab displays the effects of each option.

- Lines (page 448)
- Symbols and Arrows (page 452)
- Text (page 455)
- Fit (page 459)
- Primary Units (page 462)
- Alternate Units (page 465)
- Tolerances (page 467)

### Lines Tab

Sets the format and properties for dimension lines, extension lines, arrowheads, and center marks.




### Dimension Lines

Sets the dimension line properties.

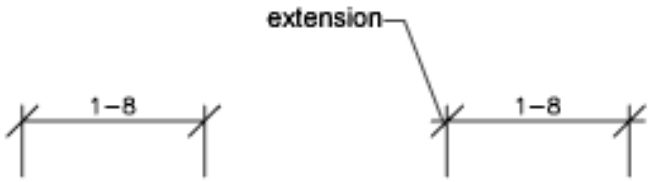
**Color** Displays and sets the color for the dimension line. If you click Select Color (at the bottom of the Color list), the Select Color dialog box (page 251) is displayed. You can also enter a color name or number. (*DIMCLRD* system variable)

You can select colors from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

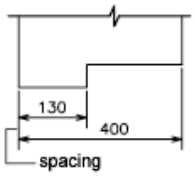
**Linetype** Sets the linetype of the dimension line. (*DIMLTYPE* system variable)

**Lineweight** Sets the lineweight of the dimension line. (*DIMLWD* system variable)

**Extend Beyond Ticks** Specifies a distance to extend the dimension line past the extension line when you use oblique, architectural, tick, integral, and no marks for arrowheads. (*DIMDLE* system variable)



**Baseline Spacing** Sets the spacing between the dimension lines of a baseline dimension. Enter a distance. For information about baseline dimensions, see *DIMBASELINE*. (*DIMDLI* system variable.)




**Suppress** Suppresses display of dimension lines. Dim Line 1 suppresses the first dimension line; Dim Line 2 suppresses the second dimension line. (*DIMSD1* and *DIMSD2* system variables)


### Extension Lines

Controls the appearance of the extension lines.

**Color** Sets the color for the extension line. If you click Select Color (at the bottom of the Color list), the Select Color dialog box (page 251) is displayed. You can also enter a color name or number. (*DIMCLRE* system variable.)

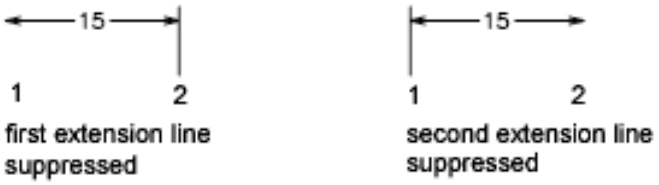
You can select colors from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

**Linetype Ext 1** Sets the linetype of the first extension line. (*DIMLTEX1* system variable)

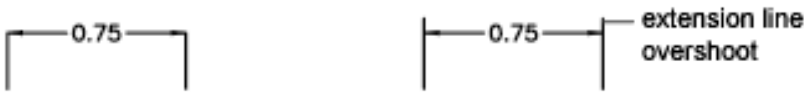
**Linetype Ext 2** Sets the linetype of the second extension line. (*DIMLTEX2* system variable)

**Lineweight** Sets the lineweight of the extension line. (*DIMLWE* system variable)

**Suppress** Suppresses the display of extension lines. Ext Line 1 suppresses the first extension line; Ext Line 2 suppresses the second extension line. (*DIMSE1* and *DIMSE2* system variables)



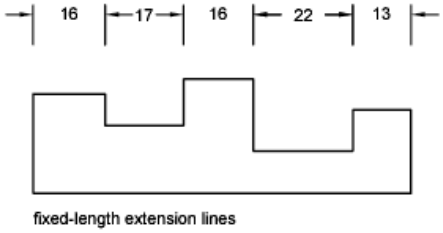

**Extend Beyond Dim Lines** Specifies a distance to extend the extension lines above the dimension line. (*DIMEXE* system variable)




**Offset From Origin** Sets the distance to offset the extension lines from the points on the drawing that define the dimension. (*DIMEXO* system variable)


**Fixed Length Extension Lines** Enables fixed length extension lines. (*DIMFXLON* system variable)

**Length** Sets the total length of the extension lines starting from the dimension line toward the dimension origin. (*DIMFXL* system variable)

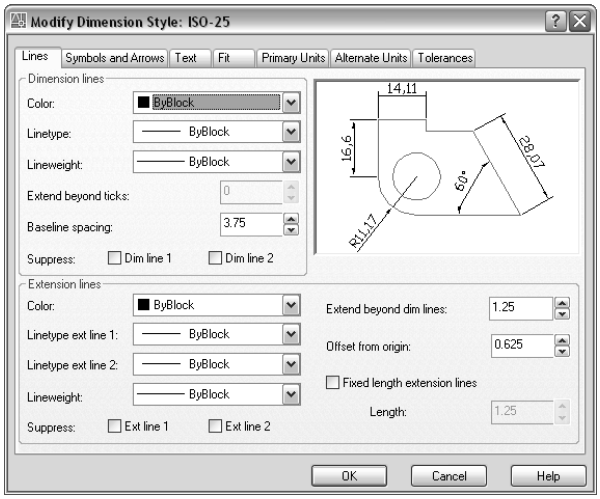



### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Symbols and Arrows Tab

Sets the format and placement for arrowheads, center marks, arc length symbols, and jogged radius dimensions.




### Arrowheads

Controls the appearance of the dimension arrowheads.

**NOTE** blocks cannot be used as custom arrowheads for dimensions or leaders.

**First** Sets the arrowhead for the first dimension line. When you change the first arrowhead type, the second arrowhead automatically changes to match it. (*DIMBLK1* system variable)

To specify a user-defined arrowhead block, select User Arrow. The Select Custom Arrow Block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing.)

**Second** Sets the arrowhead for the second dimension line. (*DIMBLK2* system variable)

To specify a user-defined arrowhead block, select User Arrow. The Select Custom Arrow Block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing.)

**Leader** Sets the arrowhead for the leader line. (*DIMLDRBLK* system variable)

To specify a user-defined arrowhead block, select User Arrow. The Select Custom Arrow Block dialog box is displayed. Select the name of a user-defined arrowhead block. (The block must be in the drawing.)

**Arrow Size** Displays and sets the size of arrowheads. (*DIMASZ* system variable)

### Center Marks

Controls the appearance of center marks and centerlines for diameter and radial dimensions. The *DIMCENTER*, *DIMDIAMETER*, and *DIMRADIUS* commands use center marks and centerlines. For DIMDIAMETER and DIMRADIUS, the center mark is drawn only if you place the dimension line outside the circle or arc.

**None** Creates no center mark or centerline. The value is stored as 0 in the DIMCEN system variable.

**Mark** Creates a center mark. The size of the center mark is stored as a positive value in the *DIMCEN* system variable.

**Line** Creates a centerline. The size of the centerline is stored as a negative value in the DIMCEN system variable.

**Size** Displays and sets the size of the center mark or centerline. (DIMCEN system variable)

### Dimension Break

Controls the gap width of dimension breaks.

**Break Size** Displays and sets the size of the gap used for dimension breaks.

### Arc Length Symbol

Controls the display of the arc symbol in an arc length dimension. (*DIMARCSYM* system variable)

**Preceding Dimension Text** Places arc length symbols before the dimension text. (*DIMARCSYM* system variable)

**Above Dimension Text** Places arc length symbols above the dimension text. (*DIMARCSYM* system variable)

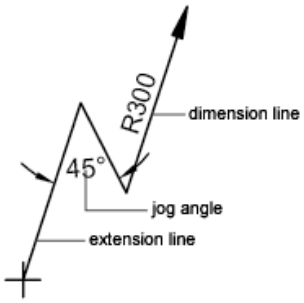
**None** Suppresses the display of arc length symbols. (*DIMARCSYM* system variable)

### Radius Jog Dimensions

Controls the display of jogged (zigzag) radius dimensions.

Jogged radius dimensions are often created when the center point of a circle or arc is located off the page.

**Jog Angle** Determines the angle of the transverse segment of the dimension line in a jogged radius dimension. (*DIMJOGANG* system variable)

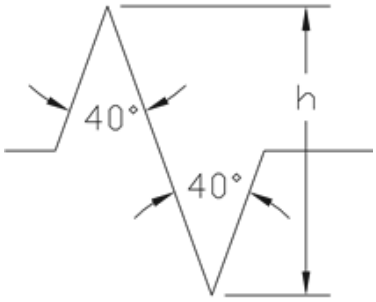



### Linear Jog Dimensions

Controls the display of the jog for linear dimensions.

Jog lines are often added to linear dimensions when the actual measurement is not accurately represent by the dimension. Typically the actual measurement is smaller than the desired value.

**Linear Jog Size** Determines the height of the of the jog, which is determined by the distance between the two vertices of the angles that make up the jog.

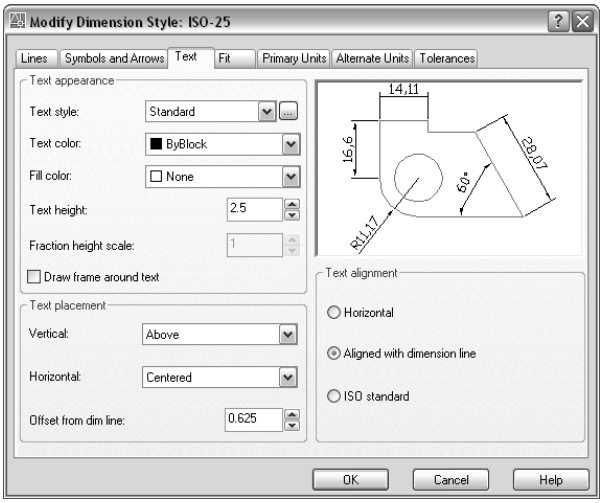


### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Text Tab

Sets the format, placement, and alignment of dimension text.




### Text Appearance

Controls the dimension text format and size.

**Text Style** Displays and sets the current style for dimension text. Select a style from the list. To create and modify styles for dimension text, choose the [...] button next to the list. (*DIMTXSTY* system variable)

**Text Style Button** Displays the Text Style dialog box (page 1367), in which you can define or modify text styles.

**Text Color** Sets the color for the dimension text. If you click Select Color (at the bottom of the Color list), the Select Color dialog box (page 251) is displayed. You can also enter color name or number. (*DIMCLRT* system variable)

You can select colors from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

**Fill Color** Sets the color for the text background in dimensions. If you click Select Color (at the bottom of the Color list), the Select Color dialog box (page

251) is displayed. You can also enter color name or number. (*DIMTFILL* and *DIMTFILLCLR* system variables)

You can select colors from the 255 AutoCAD Color Index (ACI) colors, true colors, and Color Book colors.

**Text Height** Sets the height of the current dimension text style. Enter a value in the text box. If a fixed text height is set in the Text Style (that is, the text style height is greater than 0), that height overrides the text height set here. If you want to use the height set on the Text tab, make sure the text height in the Text Style is set to 0. (*DIMTXT* system variable)

**Fraction Height Scale** Sets the scale of fractions relative to dimension text. This option is available only when Fractional is selected as the Unit Format on the Primary Units tab. The value entered here is multiplied by the text height to determine the height of dimension fractions relative to dimension text. (*DIMTFAC* system variable)

**Draw Frame Around Text** When selected, draws a frame around dimension text. Selecting this option changes the value stored in the *DIMGAP* system variable to a negative value.

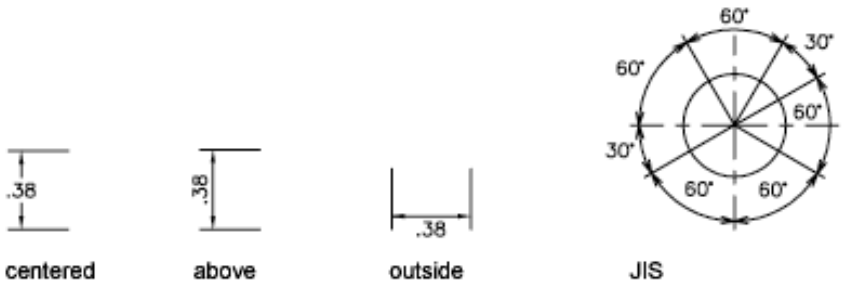
### Text Placement

Controls the placement of dimension text.

**Vertical** Controls the vertical placement of dimension text in relation to the dimension line. (*DIMTAD* system variable)

Vertical position options include the following:

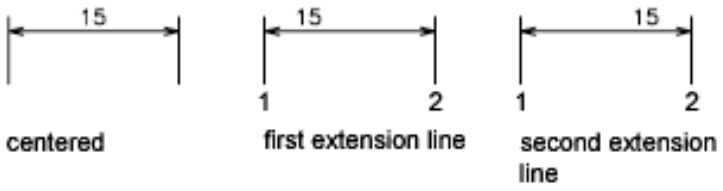
- *Centered:* Centers the dimension text between the two parts of the dimension line.
- *Above:* Places the dimension text above the dimension line. The distance from the dimension line to the baseline of the lowest line of text is the current text gap. See the Offset from Dim Line option.
- *Outside:* Places the dimension text on the side of the dimension line farthest away from the first defining point.
- *JIS:* Places the dimension text to conform to a Japanese Industrial Standards (JIS) representation.




**Horizontal** Controls the horizontal placement of dimension text along the dimension line, in relation to the extension lines. (*DIMJUST* system variable)

Horizontal position options include the following:

- *Centered:* Centers the dimension text along the dimension line between the extension lines.
- *At Ext Line 1:* Left-justifies the text with the first extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value. See Arrowheads and Offset from Dim Line.
- *At Ext Line 2:* Right-justifies the text with the second extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value. See Arrowheads and Offset from Dim Line.




- *Over Ext Line 1:* Positions the text over or along the first extension line.
- *Over Ext Line 2:* Positions the text over or along the second extension line.


**Offset from Dim Line** Sets the current text gap, which is the distance around the dimension text when the dimension line is broken to accommodate the dimension text.

This value is also used as the minimum length required for dimension line segments.

Text is positioned inside the extension lines only if the resulting segments are at least as long as the text gap. Text above or below the dimension line is placed inside only if the arrowheads, dimension text, and a margin leave enough room for the text gap. (*DIMGAP* system variable)


### Text Alignment

Controls the orientation (horizontal or aligned) of dimension text whether it is inside or outside the extension lines. (*DIMTIH* and *DIMTOH* system variables)

**Horizontal** Places text in a horizontal position.


**Aligned with Dimension Line** Aligns text with the dimension line.

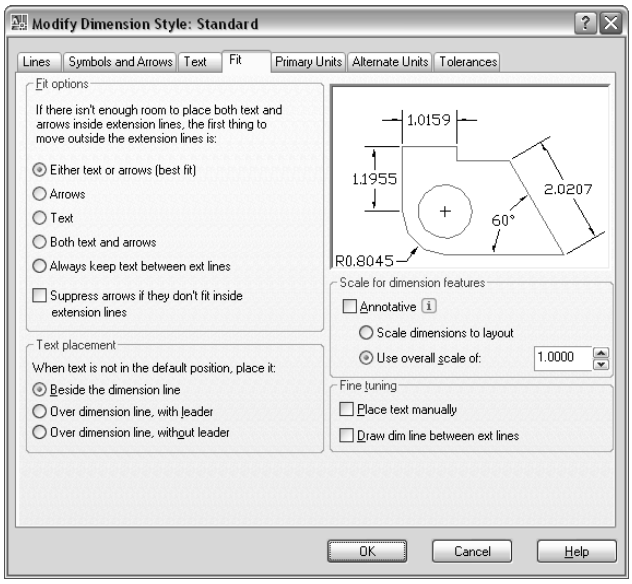
**ISO Standard** Aligns text with the dimension line when text is inside the extension lines, but aligns it horizontally when text is outside the extension lines.


### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Fit Tab

Controls the placement of dimension text, arrowheads, leader lines, and the dimension line.




### Fit Options

Controls the placement of text and arrowheads based on the space available between the extension lines.

When space is available, text and arrowheads are placed between the extension lines. Otherwise, text and arrowheads are placed according to the Fit options. (*DIMATFIT*, *DIMTIX*, and *DIMSOXD* system variables)

**Either Text or Arrows (Best Fit)** Moves either the text or the arrowheads outside the extension lines based on the best fit (*DIMATFIT* system variable).

- When enough space is available for text and arrowheads, places both between the extension lines. Otherwise, either the text or the arrowheads are moved based on the best fit.
- When enough space is available for text only, places text between the extension lines and places arrowheads outside the extension lines.
- When enough space is available for arrowheads only, places them between the extension lines and places text outside the extension lines.
- When space is available for neither text nor arrowheads, places them both outside the extension lines.

**Arrows** Moves arrowheads outside the extension lines first, then text (*DIMATFIT* system variable).

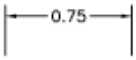
- When enough space is available for text and arrowheads, places both between the extension lines.
- When space is available for arrowheads only, places them between the extension lines and places text outside them.
- When not enough space is available for arrowheads, places both text and arrowheads outside the extension lines.

**Text** Moves text outside the extension lines first, then arrowheads (*DIMATFIT* system variable).

- When space is available for text and arrowheads, places both between the extension lines.
- When space is available for text only, places the text between the extension lines and places arrowheads outside them.
- When not enough space is available for text, places both text and arrowheads outside the extension lines.

**Both Text and Arrows** When not enough space is available for text and arrowheads, moves both outside the extension lines (*DIMATFIT* system variable).

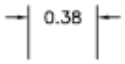

**Always Keep Text Between Ext Lines** Always places text between extension lines. (*DIMTIX* system variable)



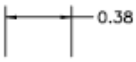

**Suppress Arrows If They Don't Fit Inside Extension Lines** Suppresses arrowheads if not enough space is available inside the extension lines. (*DIMSOXD* system variable)

### Text Placement

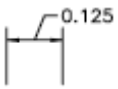
Sets the placement of dimension text when it is moved from the default position, that is, the position defined by the dimension style. (*DIMTMOVE* system variable)



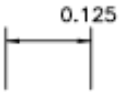

**Beside the Dimension Line** If selected, moves the dimension line whenever dimension text is moved. (*DIMTMOVE* system variable)




**Over the Dimension Line, with Leader** If selected, dimension lines are not moved when text is moved. If text is moved away from the dimension line, a leader line is created connecting the text to the dimension line. The leader line is omitted when text is too close to the dimension line. (*DIMTMOVE* system variable)




**Over the Dimension Line, Without Leader** If selected, dimension lines are not moved when text is moved. Text that is moved away from the dimension line is not connected to the dimension line with a leader. (*DIMTMOVE* system variable)




### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Scale for Dimension Features

Sets the overall dimension scale value or the paper space scaling.

**Annotative** Specifies that the dimension is . Click the information icon to learn more about annotative objects.

**Scale Dimensions To Layout** Determines a scale factor based on the scaling between the current model space viewport and paper space. (DIMSCALE system variable)

When you work in paper space, but not in a model space viewport, or when *TILEMODE* is set to 1, the default scale factor of 1.0 is used or the DIMSCALE system variable.

**Use Overall Scale Of** Sets a scale for all dimension style settings that specify size, distance, or spacing, including text and arrowhead sizes. This scale does not change dimension measurement values. (*DIMSCALE* system variable)

### Fine Tuning

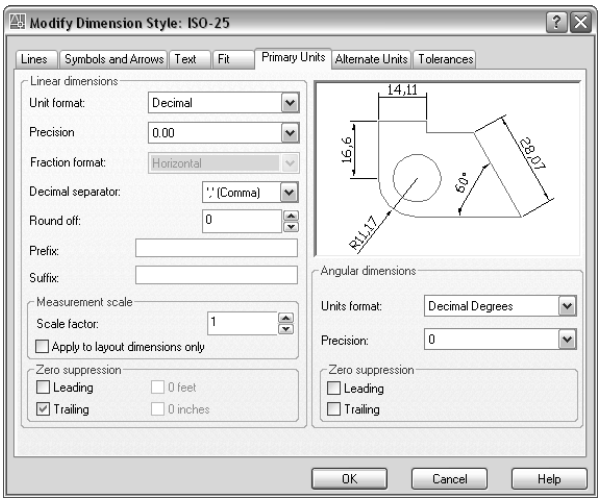
Provides additional options for placing dimension text.

**Place Text Manually** Ignores any horizontal justification settings and places the text at the position you specify at the Dimension Line Location prompt. (*DIMUPT* system variable)

**Draw Dim Line Between Ext Lines** Draws dimension lines between the measured points even when the arrowheads are placed outside the measured points. (*DIMTOFL* system variable)

### Primary Units Tab

Sets the format and precision of primary dimension units and sets prefixes and suffixes for dimension text.




### Linear Dimensions

Sets the format and precision for linear dimensions.

**Unit Format** Sets the current units format for all dimension types except Angular. (*DIMLUNIT* system variable)

The relative sizes of numbers in stacked fractions are based on the *DIMTFAC* system variable (in the same way that tolerance values use this system variable).

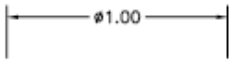
**Precision** Displays and sets the number of decimal places in the dimension text. (*DIMDEC* system variable)

**Fraction Format** Sets the format for fractions. (*DIMFRAC* system variable)

**Decimal Separator** Sets the separator for decimal formats. (*DIMDSEP* system variable)

**Round Off** Sets rounding rules for dimension measurements for all dimension types except Angular. If you enter a value of **0.25**, all distances are rounded to the nearest 0.25 unit. If you enter a value of **1.0**, all dimension distances are rounded to the nearest integer. The number of digits displayed after the decimal point depends on the Precision setting. (DIMRND system variable)

**Prefix** Includes a prefix in the dimension text. You can enter text or use control codes to display special symbols. For example, entering the control code **%%c** displays the diameter symbol. When you enter a prefix, it overrides any default prefixes such as those used in diameter and radius dimensioning. (*DIMPOST* system variable)




If you specify tolerances, the prefix is added to the tolerances as well as to the main dimension.

For more information, see Control Codes and Special Characters (page 1431).

**Suffix** Includes a suffix in the dimension text. You can enter text or use control codes to display special symbols. For example, entering the text **mm** results in dimension text similar to that shown in the illustration. When you enter a suffix, it overrides any default suffixes. (*DIMPOST* system variable)


If you specify tolerances, the suffix is added to the tolerances as well as to the main dimension.

For more information, see Control Codes and Special Characters (page 1431).

**Measurement Scale** Defines linear scale options. Applies primarily to legacy drawings.

*Scale Factor:* Sets a scale factor for linear dimension measurements. It is recommended that you do not change this value from the default value of 1.00. For example, if you enter **2**, the dimension for a 1-inch line is displayed as two inches. The value does not apply to angular dimensions and is not applied to rounding values or to plus or minus tolerance values. (*DIMLFAC* system variable)

*Apply to Layout Dimensions Only:* Applies the measurement scale factor only to dimensions created in layout viewports. Except when using nonassociative dimensions, this setting should remain unchecked. (*DIMLFAC* system variable)

**Zero Suppression** Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (*DIMZIN* system variable)

Zero suppression settings also affect real-to-string conversions performed by the AutoLISP® **rtos** and **angtos** functions.

*Leading:* Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

*Trailing:* Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

*0 Feet:* Suppresses the feet portion of a feet-and-inches dimension when the distance is less than one foot. For example, 0'-6 1/2" becomes 6 1/2".

*0 Inches:* Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

### Angular Dimensions

Displays and sets the current angle format for angular dimensions.

**Units Format** Sets the angular units format. (*DIMAUNIT* system variable)

**Precision** Sets the number of decimal places for angular dimensions. *DIMADEC* system variable)

**Zero Suppression** Controls the suppression of leading and trailing zeros. (*DIMAZIN* system variable)

*Leading:* Suppresses leading zeros in angular decimal dimensions. For example, 0.5000 becomes .5000.

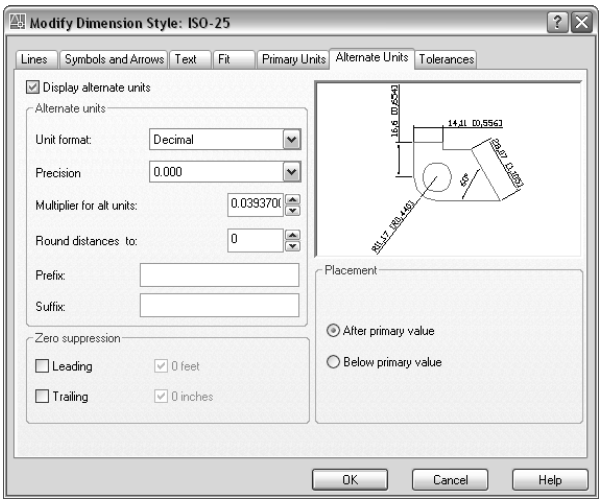
*Trailing:* Suppresses trailing zeros in angular decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Alternate Units Tab

Specifies display of alternate units in dimension measurements and sets their format and precision.



### Display Alternate Units

Adds alternate measurement units to dimension text. Sets the *DIMALT* system variable to 1.

### Alternate Units

Displays and sets the current alternate units format for all dimension types except Angular.

**Unit Format** Sets the unit format for alternate units. (*DIMALTU* system variable)

The relative sizes of numbers in stacked fractions are based on *DIMTFAC* (in the same way that tolerance values use this system variable).

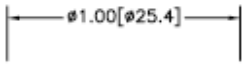
**Precision** Sets the number of decimal places for alternate units. (*DIMALTD* system variable)

**Multiplier for Alternate Units** Specifies the multiplier used as the conversion factor between primary and alternate units. For example, to convert inches to millimeters, enter **25.4**. The value has no effect on angular dimensions, and it is not applied to the rounding value or the plus or minus tolerance values. (*DIMALTF* system variable)

**Round Distances To** Sets rounding rules for alternate units for all dimension types except Angular. If you enter a value of **0.25**, all alternate measurements are rounded to the nearest 0.25 unit. If you enter a value of **1.0**, all dimension measurements are rounded to the nearest integer. The number of digits displayed after the decimal point depends on the Precision setting. (*DIMALTRND* system variable)

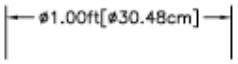
**Prefix** Includes a prefix in the alternate dimension text. You can enter text or use control codes to display special symbols. For example, entering the control code **%%c** displays the diameter symbol. (*DIMAPOST* system variable)

For more information, see Control Codes and Special Characters (page 1431).




**Suffix** Includes a suffix in the alternate dimension text. You can enter text or use control codes to display special symbols. For example, entering the text **cm** results in dimension text similar to that shown in the illustration. When you enter a suffix, it overrides any default suffixes. (*DIMAPOST* system variable)

For more information, see Control Codes and Special Characters (page 1431).




### Zero Suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (*DIMALTZ* system variable)

**Leading** Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

**Trailing** Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

**0 Feet** Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2".

**0 Inches** Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

### Placement

Controls the placement of alternate units in dimension text. (*DIMAPOST* system variable)

**After Primary Value** Places alternate units after the primary units in dimension text.

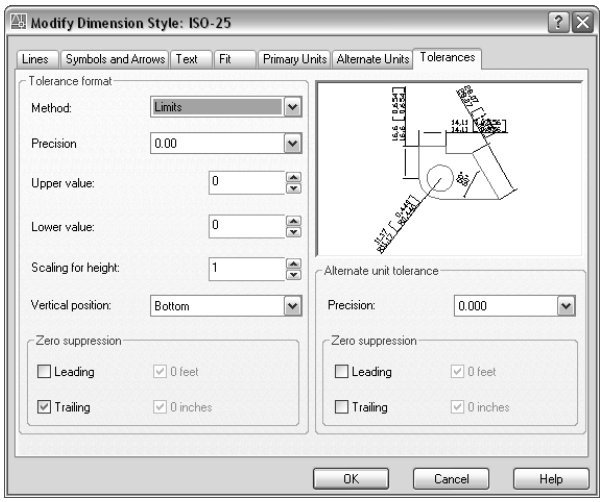
**Below Primary Value** Places alternate units below the primary units in dimension text.

### Preview

Displays sample dimension images that show the effects of changes you make to dimension style settings.

### Tolerances Tab

Controls the display and format of dimension text tolerances.

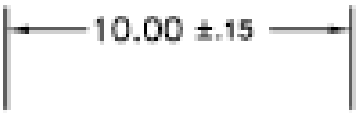



### Tolerance Format

Controls the tolerance format.

**Method** Sets the method for calculating the tolerance. (*DIMTOL* system variable)

- *None:* Does not add a tolerance. The *DIMTOL* system variable is set to 0.
- *Symmetrical:* Adds a plus/minus expression of tolerance in which a single value of variation is applied to the dimension measurement. A plus-or-minus sign appears after the dimension. Enter the tolerance value in Upper Value. The *DIMTOL* system variable is set to 1. The *DIMLIM* system variable is set to 0.




- *Deviation:* Adds a plus/minus tolerance expression. Different plus and minus values of variation are applied to the dimension measurement. A plus sign (+) precedes the tolerance value entered in Upper Value, and a minus sign (-) precedes the tolerance value entered in Lower Value. The *DIMTOL* system variable is set to 1. The *DIMLIM* system variable is set to 0.


- *Limits:* Creates a limit dimension. A maximum and a minimum value are displayed, one over the other. The maximum value is the dimension value plus the value entered in Upper Value. The minimum value is the dimension value minus the value entered in Lower Value. The *DIMTOL* system variable is set to 0. The *DIMLIM* system variable is set to 1.


- *Basic:* Creates a basic dimension, which displays a box around the full extents of the dimension. The distance between the text and the box is stored as a negative value in the *DIMGAP* system variable.


**Precision** Sets the number of decimal places. (*DIMTDEC* system variable)

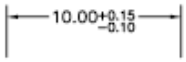
**Upper Value** Sets the maximum or upper tolerance value. When you select Symmetrical in Method, this value is used for the tolerance. (*DIMTP* system variable)

**Lower Value** Sets the minimum or lower tolerance value. (*DIMTM* system variable)

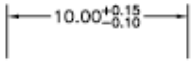
**Scaling for Height** Sets the current height for the tolerance text. The ratio of the tolerance height to the main dimension text height is calculated and stored in the *DIMTFAC* system variable.

**Vertical Position** Controls text justification for symmetrical and deviation tolerances.

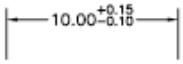
- *Top:* Aligns the tolerance text with the top of the main dimension text. When you select this option, the *DIMTOLJ* system variable is set to 2.




- *Middle:* Aligns the tolerance text with the middle of the main dimension text. When you select this option, the DIMTOLJ system variable is set to 1.




- *Bottom:* Aligns the tolerance text with the bottom of the main dimension text. When you select this option, the DIMTOLJ system variable is set to 0.




### Tolerance Alignment

Controls the alignment of upper and lower tolerance values when stacked

**Align Decimal Separators** Values are stacked by their decimal separators.

**Align Operational Symbols** Values are stacked by their operational symbols.

### Zero Suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (*DIMTZIN* system variable)

Zero suppression settings also affect real-to-string conversions performed by the AutoLISP® **rtos** and **angtos** functions.

**Leading** Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

**Trailing** Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

**0 Feet** Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2".

**0 Inches** Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

#### Alternate Unit Tolerance

Formats alternate tolerance units.

**Precision** Displays and sets the number of decimal places. (*DIMALTTD* system variable)

#### Zero Suppression

Controls the suppression of leading and trailing zeros and of feet and inches that have a value of zero. (*DIMALTTZ* system variable)

**Leading** Suppresses leading zeros in all decimal dimensions. For example, 0.5000 becomes .5000.

**Trailing** Suppresses trailing zeros in all decimal dimensions. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

**0 Feet** Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2".

**0 Inches** Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

#### Preview

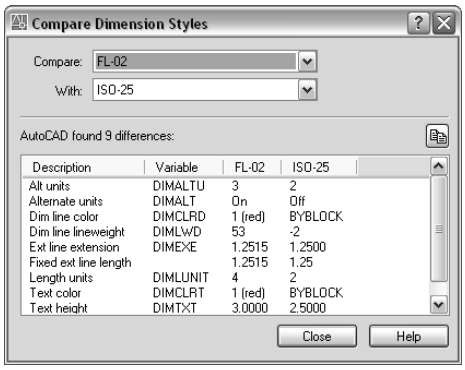
Displays sample dimension images that show the effects of changes you make to dimension style settings.

## Compare Dimension Styles Dialog Box

### Quick Reference

Styles 
Format ➤ Dimension StyleAt the Command prompt, enter dimstyle.
Dimension ➤ Dimension StyleAt the Command prompt, enter dimstyle.
**dimstyle**

Compares the properties of two dimension styles or displays all properties of one style. You can print the results of the comparison to the Clipboard, and then paste to other MicrosoftWindowsapplications.




**Compare** Specifies the first dimension style for the comparison.

**With** Specifies the second dimension style for the comparison. If you set the second style to <none> or to the same style as the first, all the properties of the dimension style are displayed.

Comparison results are displayed automatically under the following headings:

- Description of the dimension style property
- System variable that controls the property
- System variable values of style properties that differ for each dimension style

**Results** Displays the results of the dimension style comparison. If you compare two different styles, the properties that differ are displayed. If you set the second style to <none> or to the same style as the first, all the properties of the dimension style are displayed.

**Print to Clipboard button** Copies the results of the comparison to the Clipboard. You can then paste the results to other Windows applications, such as word processors and spreadsheets.

# -DIMSTYLE

## Quick Reference

If you enter **-dimstyle** at the command prompt, the following DIMSTYLE command prompts are displayed.

Current dimension style: <*current*> Annotative: <*current*>
Enter a dimension style option
[ANnotative (page 473)/Save (page 473)/Restore (page 474)/STatus (page 475)/Variables (page 475)/Apply (page 475)/? (page 476)] <Restore>: *Enter an option or press* ENTER

### Annotative

Creates an dimension style.

Create annotative dimension style [Yes/No] <Yes>: *Enter* **y** *or* **n** *or press* ENTER

### Save

Saves the current settings of dimensioning system variables to a dimension style.

Enter name for new dimension style or [?]: *Enter a name or enter* **?**

### Name

Saves the current settings of dimensioning system variables to a new dimension style using the name you enter. The new dimension style becomes the current one.

If you enter the name of an existing dimension style, the following prompts are displayed:

That name is already in use, redefine it? <N>: *Enter* **y** *or press* ENTER

If you enter **y**, associative dimensions that use the redefined dimension style are regenerated.

To display the differences between the dimension style name you want to save and the current style, enter a tilde (~) followed by the style name at the Enter Name for New Dimension Style prompt. Only settings that differ are displayed, with the current setting in the first column, and the setting of the

compared style in the second column. After the differences are displayed, the previous prompt returns.

#### ?—List Dimension Styles

Lists the named dimension styles in the current drawing.

Enter dimension style(s) to list <*>: *Enter a name, a partial name with wild-card characters, or press* ENTER *to list all dimension styles*

After the named dimension styles are listed, the previous prompt returns.

### Restore

Restores dimensioning system variable settings to those of a selected dimension style.

Enter dimension style name, [?] or <select dimension>: *Enter a name, enter* **?**, *or press* ENTER *to select a dimension*

### Name

Makes the dimension style you enter the current dimension style.

To display the differences between the dimension style name you want to restore and the current style, enter a tilde (~) followed by the style name at the Enter Dimension Style Name prompt. Only settings that differ are displayed, with the current setting in the first column, and the setting of the compared style in the second column. After the differences are displayed, the previous prompt returns.

#### ?—List Dimension Styles

Lists the named dimension styles in the current drawing.

Enter dimension style(s) to list <*>: *Enter a name list or press* ENTER

After the dimension styles are listed, the previous prompt returns.

### Select Dimension

Makes the dimension style of the selected object the current dimension style.

Select dimension:

### Status

Displays the current values of all dimension system variables. After listing the variables, DIMSTYLE ends.

### Variables

Lists the dimension system variable settings of a dimension style or selected dimensions without modifying the current settings.

Enter a dimension style name, [?] or <select dimension>: *Enter a name, enter* **?***, or press* ENTER *to select dimensions*

### Name

Lists the settings of dimension system variables for the dimension style name you enter. After listing the variables, DIMSTYLE ends.

To display the differences between a particular dimension style and the current style, enter a tilde (~) followed by the style name at the Enter Dimension Style Name prompt. Only settings that differ are displayed, with the current setting in the first column, and the setting of the compared style in the second column. After the differences are displayed, the previous prompt returns.

### ?—List Dimension Styles

Lists the named dimension styles in the current drawing.

Enter dimension style(s) to list <*>: *Enter a name list or press* ENTER

After the dimension styles are listed, the previous prompt returns.

### Select Dimension

Lists the dimension style and any dimension overrides for the dimension object you select.

Select dimension:

### Apply

Applies the current dimensioning system variable settings to selected dimension objects, permanently overriding any existing dimension styles applied to these objects.

Select objects: *Use an object selection method to select a dimension object*

The dimension line spacing between existing baseline dimensions is not updated (see the *DIMDLI* system variable); dimension text variable settings do not update existing leader text.

#### ?—List Dimension Styles

Lists the named dimension styles in the current drawing.

Enter dimension style(s) to list <*>: *Enter a name, a partial name with wild-card characters, or press* ENTER

# DIMTEDIT

## Quick Reference

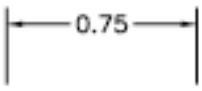
Moves and rotates dimension text

Dimension 
Dimension ➤ Align Text
**dimtedit**
Select dimension: *Select a dimension object*

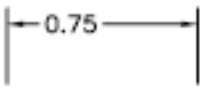
You are prompted for the new location of the dimension text.

Specify new location for dimension text (page 476) or [Left (page 476)/Right (page 477)/Center (page 477)/Home (page 477)/Angle (page 477)]: *Specify a point or enter an option*
**Location for Dimension Text** Updates the location of the dimension text dynamically as you drag it. To determine whether text appears above, below, or in the middle of the dimension line, use the Text tab in the New, Modify, and Override Dimension Style dialog box.

**Left** Left-justifies the dimension text along the dimension line. This option works only with linear, radial, and diameter dimensions.

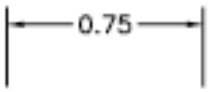


before Left

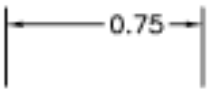


after Left

**Right** Right-justifies the dimension text along the dimension line. This option works only with linear, radial, and diameter dimensions.



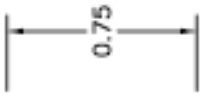

before Right



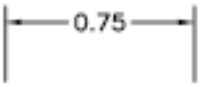

after Right

**Center** Centers the dimension text on the dimension line.

**Home** Moves the dimension text back to its default position.



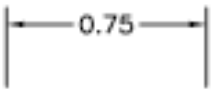

before Home



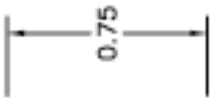

after Home

**Angle** Changes the angle of the dimension text.

Enter text angle:




before Angle




after Angle 90 deg

The center point of the text does not change. If the text moves or the dimension is regenerated, the orientation set by the text angle is retained. Entering an angle of 0 degrees puts the text in its default orientation.

# DIST

## Quick Reference

Measures the distance and angle between two points

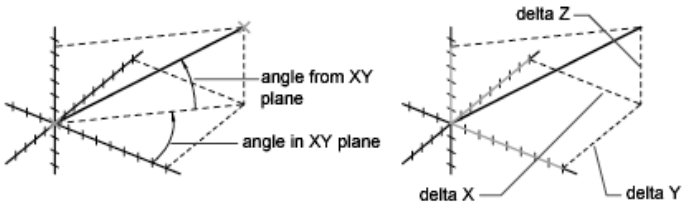
Inquiry 

Tools ➤ Inquiry ➤ DistanceAt the Command prompt, enter dist.

**dist** (or '**dist** for transparent use)
Specify first point: *Specify a point*
Specify second point: *Specify a point*
Distance = *calculated distance*, Angle in XY plane = *angle*,
Angle from XY plane = *angle*
Delta X = *change in* X, Delta Y = *change in* Y, Delta Z = *change in* Z

The true 3D distance between points is reported. The angle in the *XY* plane is relative to the current *X* axis. The angle from the *XY* plane is relative to the current *XY* plane. DIST assumes the current elevation for the first or second point if you omit the *Z* coordinate value.

The distance is displayed using the current units format.




**NOTE** When you work on a layout in paper space, distances are normally reported in paper space units. However, if you use the DIST command with object snaps on model space objects that are displayed *within a single viewport*, distances are reported in 2D model space units. When using the DIST command for 3D distances, it is recommended that you switch to model space.

# DISTANTLIGHT

## Quick Reference

Creates a distant light

Lights 
View ➤ Render ➤ Light ➤ New Distant LightAt the Command prompt, enter distantlight.
**distantlight**
Lights panel (click to expand), Create a Distant Light
Specify light direction FROM <0,0,0> or [Vector]: *Specify a point or enter***v**

Specify light direction TO <1,1,1> *Specify a point*

If you enter the Vector option, the following prompt is displayed:

Specify vector direction <0.0000,-0.0100,1.0000>: *Enter a vector*

After you specify the light direction and if the LIGHTINGUNITS system variable is set to 0, the following prompt is displayed:

Enter an option to change [Name (page 479)/Intensity (page 479)/Status (page 479)/shadoW (page 481)/Color (page 481)/eXit (page 481)] <eXit>:

If the LIGHTINGUNITS system variable is set to 1 or 2, the following prompt is displayed:

Enter an option to change [Name (page 479)/Intensity factor (page 479)/Status (page 479)/Photometry (page 480)/shadoW (page 481)/filterColor (page 481)/eXit (page 481)] <eXit>:

---

**NOTE** When the LIGHTINGUNITS system variable is set to 1 or 2, the Attenuation option has no affect on the creation of the light. It is only maintained for scripting compatibility.

---

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity/Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00 - max float) <1.0000>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf] <On>:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived power per unit of solid angle. The total luminous flux is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i**, you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*> :

Enter a text string using wild card characters to display a partial listing of color names, or an asterick (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>:
**Off** Turns off display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory that should be used to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness to use to calculate the shadow map.

### Color/Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]<255,255,255>:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L) <0,0,100>:

**Color Book** Specifies a color from a color book.

Enter Color Book name:

### Exit

Exits the command.

# DIVIDE

## Quick Reference

Places evenly spaced point objects or blocks along the length or perimeter of an object

Draw ➤ Point ➤ DivideAt the Command prompt, enter divide.
**divide**
Select object to divide: *Use an object selection method*
Enter number of segments (page 482) or [Block (page 482)]: *Enter a value from 2 through 32,767, or enter* **b**

### Number of Segments

Places point objects at equal intervals along the selected objects.

The illustration shows a polyline divided into five parts. Point Display mode (*PDMODE*) has been set such that the points can be seen.


### Block

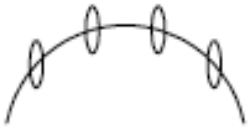
Places blocks at equal intervals along the selected object.

Enter name of block to insert: *Enter the name of a block currently defined in the drawing*
Align block with object? [Yes/No] <Y>: *Enter* **y** *or* **n** *or press* ENTER
**Yes** Specifies that the *X* axes of the inserted blocks be tangent to, or collinear with, the divided object at the dividing points.

**No** Aligns the blocks according to their normal orientation.

Enter number of segments: *Enter a value from 2 through 32,767*

The illustration shows an arc divided into five equal parts using a block consisting of a vertically oriented ellipse.



block not aligned

block aligned

# DONUT

## Quick Reference

Draws filled circles and rings

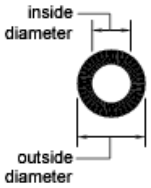
Draw ➤ DonutAt the Command prompt, enter donut.

**donut**

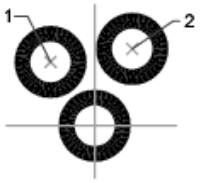
Specify inside diameter of donut <*current*>: *Specify a distance or press* ENTER

If you specify an inside diameter of 0, the donut is a filled circle.

Specify outside diameter of donut <*current*>: *Specify a distance or press* ENTER
Specify center of donut or <exit>: *Specify a point (1) or press* ENTER *to end the command*




The location of the donut is set based on the center point. After you specify the diameters, you are prompted for the locations at which to draw donuts. A donut is drawn at each point specified (2). How the interior of a donut is filled depends on the current setting of the *FILL* command.



# DRAGMODE

## Quick Reference

Controls the way dragged objects are displayed

**dragmode** (or '**dragmode** for transparent use)
Enter new value [ON (page 484)/OFF (page 484)/Auto (page 484)] <*current*>: *Enter an option or press* ENTER
**On** Permits dragging, but you must enter **drag** where appropriate in a drawing or editing command to initiate dragging.


**Off** Ignores all dragging requests, including those embedded in menu items.


**Auto** Turns on dragging for every command that supports it. Dragging is performed whenever it is possible. Entering **drag** each time is not necessary.

# DRAWINGRECOVERY

## Quick Reference

Displays a list of drawing files that can be recovered after a program or system failure

File ➤ Drawing Utilities ➤ Drawing RecoveryDoes not exist in the menus.
**drawingrecovery**

Opens the Drawing Recovery Manager (page 485).

# Drawing Recovery Manager

## Quick Reference

File ➤ Drawing Utilities ➤ Drawing RecoveryDoes not exist in the menus.
**drawingrecovery**

The Drawing Recovery Manager displays a list of all drawing files that were open at the time of a program or system failure. You can preview and open each drawing or backup file to choose which one should be saved as the primary DWG file.

### Backup Files

Displays the drawings that may need to be recovered after a program or system failure. A top-level drawing node contains a set of files associated with each drawing. You can expand a top-level node to display the drawing files and backup files that are candidates for recovery. If available, up to four files are displayed including

- The recovered drawing file saved at the time of a program failure (DWG, DWS)
- The automatic save file, also called the "autosave" file (SV$)
- The drawing backup file (BAK)
- The original drawing file (DWG, DWS)

Once a drawing or backup file is opened and saved, the corresponding top-level drawing node is removed from the Backup Files area.

### Details

Provides the following information about the currently selected node in the Backup Files area:

- When a top-level drawing node is selected, information about each available drawing or backup file associated with the original drawing is displayed.
- When an individual drawing or backup file is selected, additional information about that file is displayed.

#### Preview

Displays a thumbnail preview of the currently selected drawing or backup file.

#### Shortcut Menu Options

Right-click a drawing node, drawing or backup file, or a blank area in the Backup Files area to display a shortcut menu with relevant options.

**Open All** Opens all the drawing and backup files associated with the selected, top-level drawing node.

**Remove** Removes the selected, top-level drawing node.

**Open** Opens the selected drawing or backup file for drawing recovery. You can select multiple files using SHIFT and CTRL.

**Properties** Displays the File Properties dialog box from Windows Explorer for the selected drawing or backup file.

**Expand All** Expands all top-level drawing nodes. Access this option by right-clicking a blank area in the Backup Files area.

**Collapse All** Collapses all top-level drawing nodes. Access this option by right-clicking a blank area in the Backup Files area.

# DRAWINGRECOVERYHIDE

## Quick Reference

Closes the Drawing Recovery Manager

**drawingrecoveryhide**

Closes the Drawing Recovery Manager.

# DRAWORDER

## Quick Reference

Changes the draw order of images and other objects

Draw Order 
Tools ➤ Draw Order
Select an object, right-click, and then click Draw Order.
**draworder**
Select objects: *Use an object selection method*
Enter object ordering option [Above objects (page 487)/Under objects (page 487)/Front (page 487)/Back (page 487)] <Back>: *Enter an option or press* ENTER
**Above Objects** Moves the selected object above the specified reference objects.

Select reference objects: *Use an object selection method*

**Under Objects** Moves the selected objects below the specified reference objects.

Select reference objects: *Use an object selection method*

**Front** Moves the selected objects to the top of the order of objects in the drawing.

**Back** Moves the selected objects to the bottom of the order of objects in the drawing.

When you change the draw order (display and plotting order) of multiple objects, the relative draw order of the selected objects is maintained.

By default, when you create new objects from existing ones (for example, FILLET or PEDIT), the new objects are assigned the draw order of the original object you selected first. By default, while you edit an object (for example, MOVE or STRETCH), the object is displayed on top of all objects in the drawing. When you are finished editing, your drawing is partially regenerated so that the object is displayed according to its correct draw order. This can result in some edit operations taking slightly longer. You can use *DRAWORDERCTL* to change the default draw order settings. *TEXTTOFRONT* changes the draw order of all text and dimensions in the drawing.

# DSETTINGS

## Quick Reference

Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input

Tools ➤ Drafting SettingsAt the Command prompt, enter dsettings.
Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar. Click Settings.
**dsettings** (or '**dsettings** for transparent use)

The Drafting Settings dialog box (page 488) is displayed.

# Drafting Settings Dialog Box

## Quick Reference

Tools ➤ Drafting Settings
Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar and click Settings.
**dsettings** (or '**dsettings** for transparent use)

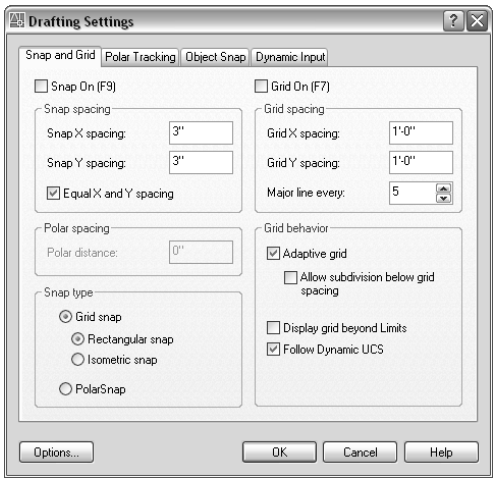
Specifies drafting settings organized for drawing aids in four categories: Snap and Grid (page 488), Polar Tracking (page 491), Object Snap (page 492), and Dynamic Input (page 496).

### Options

Displays the Drafting tab in the Options dialog box (page 946). You cannot access the Options dialog box from the Drafting Settings dialog box if you are running DSETTINGS transparently.

### Snap and Grid Tab (Drafting Settings Dialog Box)

Specifies Snap and Grid settings.



### Snap On

Turns Snap mode on or off. You can also turn Snap mode on or off by clicking Snap on the status bar, by pressing F9, or by using the *SNAPMODE* system variable.

### Snap Spacing

Controls an invisible, rectangular grid of snap locations that restricts cursor movement to specified *X* and *Y* intervals.

**Snap X Spacing** Specifies the snap spacing in the *X* direction. The value must be a positive real number. (*SNAPUNIT* system variable)

**Snap Y Spacing** Specifies the snap spacing in the *Y* direction. The value must be a positive real number. (*SNAPUNIT* system variable)

**Equal X and Y Spacing** Forces the X and Y spacing to the same values for snap spacing and for grid spacing. The snap spacing intervals can be different from the grid spacing intervals.

### Polar Spacing

Controls the PolarSnap™ increment distance.

**Polar Distance** Sets the snap increment distance when PolarSnap is selected under Snap Type & Style. If this value is 0, the PolarSnap distance assumes the value for Snap X Spacing. The Polar Distance setting is used in conjunction with polar tracking and/or object snap tracking. If neither tracking feature is enabled, the Polar Distance setting has no effect. (*POLARDIST* system variable)

### Snap Type

Sets the snap style and snap type.

**Grid Snap** Sets the snap type to Grid. When you specify points, the cursor snaps along vertical or horizontal grid points. (*SNAPTYPE* system variable)

*Rectangular Snap:* Sets the snap style to standard Rectangular snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to a rectangular snap grid. (*SNAPSTYL* system variable)

*Isometric Snap:* Sets the snap style to Isometric snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to an isometric snap grid. (*SNAPSTYL* system variable)

**PolarSnap** Sets the snap type to Polar. When Snap mode is on and you specify points with polar tracking turned on, the cursor snaps along polar alignment

angles set on the Polar Tracking tab relative to the starting polar tracking point. (*SNAPTYPE* system variable)

### Grid On

Turns the grid on or off. You can also turn grid mode on or off by clicking Grid on the status bar, by pressing F7, or by using the *GRIDMODE* system variable.

### Grid Spacing

Controls the display of a grid that helps you visualize distances.

---

**NOTE** The limits of the grid are controlled by the *LIMITS* command and the *GRIDDISPLAY* system variable.

---

**Grid X Spacing** Specifies the grid spacing in the *X* direction. If this value is 0, the grid assumes the value set for Snap X Spacing. (*GRIDUNIT* system variable)

**Grid Y Spacing** Specifies the grid spacing in the *Y* direction. If this value is 0, the grid assumes the value set for Snap Y Spacing. (*GRIDUNIT* system variable)

**Major Line Every** Specifies the frequency of major grid lines compared to minor grid lines. Grid lines rather than grid dots are displayed when the *VSCURRENT* is set to any visual style except 2D Wireframe. (*GRIDMAJOR* system variable)

### Grid Behavior

Controls the appearance of the grid lines that are displayed when *VSCURRENT* is set to any visual style except 2D Wireframe.

**Adaptive Grid** Limits the density of the grid when zoomed out. (*GRIDDISPLAY* system variable)

*Allow Subdivision Below Grid Spacing:*

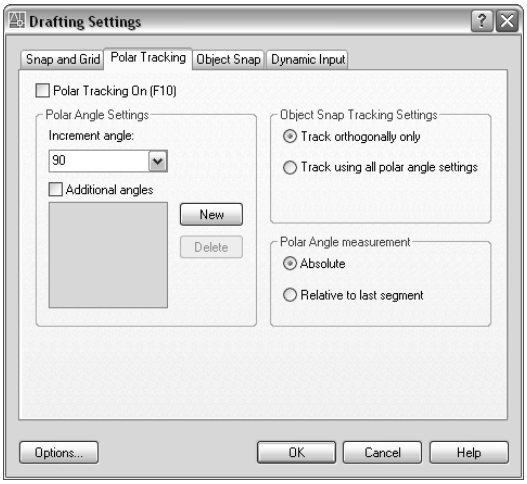
Generates additional, more closely spaced grid lines when zoomed in. The frequency of these grid lines is determined by the frequency of the major grid lines. (*GRIDDISPLAY* and *GRIDMAJOR* system variables)

**Display Grid Beyond Limits** Displays the grid beyond the area specified by the *LIMITS* command. (*GRIDDISPLAY* system variable)

**Follow Dynamic UCS** Changes the grid plane to follow the XY plane of the dynamic UCS. (*GRIDDISPLAY* system variable)

### Polar Tracking Tab (Drafting Settings Dialog Box)

Controls the AutoTrack settings.




#### Polar Tracking On

Turns polar tracking on and off. You can also turn polar tracking on or off by pressing F10 or by using the *AUTOSNAP* system variable.

#### Polar Angle Settings

Sets the alignment angles for polar tracking. (*POLARANG* system variable)

**Increment Angle** Sets the polar increment angle used to display polar tracking alignment paths. You can enter any angle, or select a common angle of 90, 45, 30, 22.5, 18, 15, 10, or 5 degrees from the list. (*POLARANG* system variable)

**Additional Angles** Makes any additional angles in the list available for polar tracking. The Additional Angles check box is also controlled by the *POLARMODE* system variable, and the list of additional angles is also controlled by the *POLARADDANG* system variable.

---

**NOTE** Additional angles are absolute, not incremental.

---

**List of Angles** If Additional Angles is selected, lists the additional angles that are available. To add new angles, click New. To remove existing angles, click Delete. (POLARADDANG system variable)

**New** Adds up to 10 additional polar tracking alignment angles.

**NOTE** Before adding fractional angles, you must set the *AUPREC* system variable to the appropriate decimal precision to avoid undesired rounding. For example, if the value of AUPREC is 0 (the default value), all fractional angles you enter are rounded to the nearest whole number.

**Delete** Deletes selected additional angles.

### Object Snap Tracking Settings

Sets options for object snap tracking.

**Track Orthogonally Only** Displays only orthogonal (horizontal/vertical) object snap tracking paths for acquired object snap points when object snap tracking is on. (POLARMODE system variable)

**Track Using All Polar Angle Settings** Applies polar tracking settings to object snap tracking. When you use object snap tracking, the cursor tracks along polar alignment angles from acquired object snap points. (POLARMODE system variable)

**NOTE** Clicking Polar and Otrack on the status bar also turns polar tracking and object snap tracking on and off.

### Polar Angle Measurement

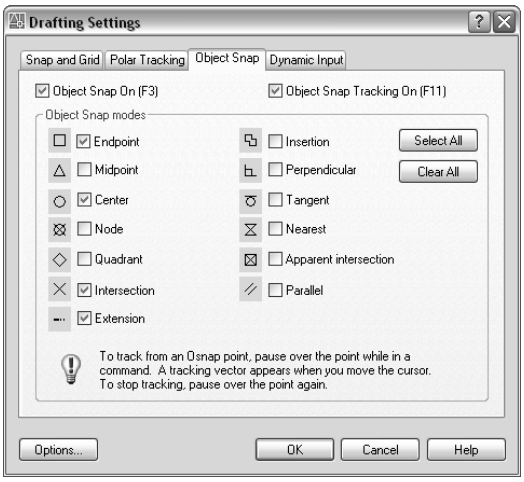
Sets the basis by which polar tracking alignment angles are measured.

**Absolute** Bases polar tracking angles on the current user coordinate system (UCS).

**Relative to Last Segment** Bases polar tracking angles on the last segment drawn.

### Object Snap Tab (Drafting Settings Dialog Box)

Controls running object snap settings. With running object snap settings, also called Osnap, you can specify a snap point at an exact location on an object. When more than one option is selected, the selected snap modes are applied to return a point closest to the center of the aperture box. Press TAB to cycle through the options.




### Object Snap On

Turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. (*OSMODE* system variable)

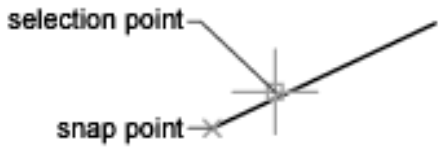
### Object Snap Tracking On

Turns object snap tracking on and off. With object snap tracking, the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps. (*AUTOSNAP* system variable)

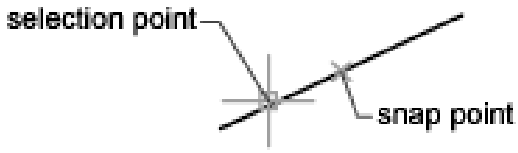
### Object Snap Modes

Lists object snaps that you can turn on as running object snaps.

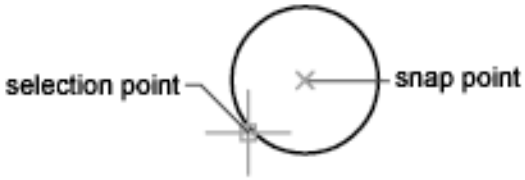
**Endpoint** Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray, or to the closest corner of a trace, solid, or 3D face.



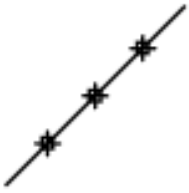

**Midpoint** Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, region, solid, spline, or xline.



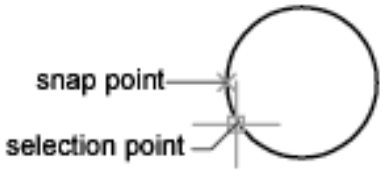

**Center** Snaps to the center of an arc, circle, ellipse, or elliptical arc.




**Node** Snaps to a point object, dimension definition point, or dimension text origin.



**Quadrant** Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.




**Intersection** Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, spline, or xline. Extended Intersection is not available as a running object snap.

Intersection and Extended Intersection do not work with edges or corners of 3D solids.

**NOTE** You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

**Extension** Causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension.

**NOTE** When working in perspective view, you cannot track along the extension line of an arc or elliptical arc.

**Insertion** Snaps to the insertion point of an attribute, a block, a shape, or text.

**Perpendicular** Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, solid, spline, or xline. Deferred Perpendicular snap mode is automatically turned on when the object you are drawing requires that you complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, an AutoSnap tooltip and marker are displayed.


**Tangent** Snaps to the tangent of an arc, circle, ellipse, elliptical arc, or spline. Deferred Tangent snap mode is automatically turned on when the object you are drawing requires that you complete more than one tangent snap. You can use Deferred Tangent to draw a line or xline that is tangent to arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, a marker and an AutoSnap tooltip are displayed.


**NOTE** When you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.

**Nearest** Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, ray, spline, or xline.

**Apparent Intersection** Snaps to the visual intersection of two objects that are not in the same plane but may appear to intersect in the current view. Extended Apparent Intersection is not available as a running object snap. Apparent and Extended Apparent Intersection do not work with edges or corners of 3D solids.

**NOTE** You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

**Parallel** Constrains a line segment, polyline segment, ray or xline to be parallel to another linear object. After you specify the first point of a linear object, specify the parallel object snap. Unlike other object snap modes, you move the cursor and *hover* over another linear object until the angle is acquired. Then, move the cursor back toward the object that you are creating. When the path of the object is parallel to the previous linear object, an alignment path is displayed, which you can use to create the parallel object.

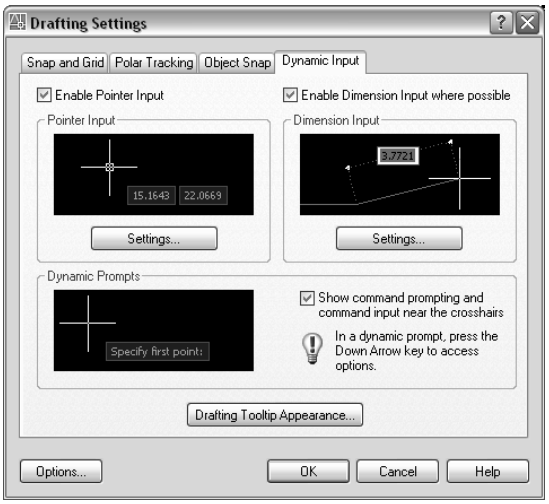
**NOTE** Turn off ORTHO mode before using the parallel object snap. Object snap tracking and polar snap are turned off automatically during a parallel object snap operation. You must specify the first point of a linear object before using the parallel object snap.

**Select All** Turns on all object snap modes.

**Clear All** Turns off all object snap modes.

### Dynamic Input Tab (Drafting Settings Dialog Box)

Controls pointer input, dimension input, dynamic prompting, and the appearance of drafting tooltips.




### Enable Pointer Input

Turns on pointer input. When pointer input and dimensional input are both turned on, dimensional input supersedes pointer input when it is available. (*DYNMODE* system variable)

### Pointer Input

Displays the location of the crosshairs as coordinate values in a tooltip near the cursor. When a command prompts for a point, you can enter coordinate values in the tooltip instead of on the command line.

**Preview Area** Shows an example of pointer input.

**Settings** Displays the Pointer Input Settings dialog box (page 498).

### Enable Dimension Input

Turns on dimensional input. Dimensional input is not available for some commands that prompt for a second point. (*DYNMODE* system variable)

### Dimension Input

Displays a dimension with tooltips for distance value and angle value when a command prompts for a second point or a distance. The values in the dimension tooltips change as you move the cursor. You can enter values in the tooltip instead of on the command line.

**Preview Area** Shows an example of dimensional input.

**Settings** Displays the Dimension Input Settings dialog box (page 499).

#### Dynamic Prompts

Displays prompts in a tooltip near the cursor when necessary in order to complete the command. You can enter values in the tooltip instead of on the command line.

**Preview Area** Shows an example of dynamic prompts.

**Show Command Prompting and Command Input near the Crosshairs** Displays prompts in Dynamic Input tooltips. (*DYNPROMPT* system variable)

#### Drafting Tooltip Appearance

Displays the Tooltip Appearance dialog box. (page 500)

## Pointer Input Settings Dialog Box

### Quick Reference

Tools ➤ Drafting Settings
Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar and click Settings.
**dsettings** (or '**dsettings** for transparent use)

#### Format

Controls coordinate format in the tooltips that are displayed when pointer input is turned on.

**Polar Format** Displays the tooltip for the second or next point in polar coordinate format. Enter a comma (,) to change to Cartesian format. (*DYNPIFORMAT* system variable)

**Cartesian Format** Displays the tooltip for the second or next point in Cartesian coordinate format. Enter an angle symbol (<) to change to polar format. (*DYNPIFORMAT* system variable)

**Relative Coordinates** Displays the tooltip for the second or next point in relative coordinate format. Enter a pound sign (#) to change to absolute format. (*DYNPICOORDS* system variable)

**Absolute Coordinates** Displays the tooltip for the second or next point in absolute coordinate format. Enter an at sign (@) to change to relative format. Note that you cannot use the direct distance method when this option is selected. (*DYNPICOORDS* system variable)

#### Visibility

Controls when pointer input is displayed. (*DYNPIVIS* system variable)

**As Soon As I Type Coordinate Data** When pointer input is turned on, displays tooltips only when you start to enter coordinate data. (*DYNPIVIS* system variable)

**When a Command Asks for a Point** When pointer input is turned on, displays tooltips whenever a command prompts for a point. (*DYNPIVIS* system variable)

**Always—Even When Not in a Command** Always displays tooltips when pointer input is turned on. (*DYNPIVIS* system variable)

## Dimension Input Settings Dialog Box

### Quick Reference

Tools ➤ Drafting Settings
Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar and click Settings.
**dsettings** (or '**dsettings** for transparent use)

#### Visibility

Controls which tooltips are displayed during grip stretching when dimensional input is turned on. (*DYNDIVIS* system variable)

**Show Only 1 Dimension Input Field at a Time** Displays only the length change dimensional input tooltip when you are using grip editing to stretch an object. (*DYNDIVIS* system variable)

**Show 2 Dimension Input Fields at a Time** Displays the length change and resulting dimensional input tooltips when you are using grip editing to stretch an object. (*DYNDIVIS* system variable)

**Show the Following Dimension Input Fields Simultaneously** When you are using grip editing to stretch an object, displays the dimensional input tooltips that are selected below. (*DYNDIVIS* and *DYNDIGRIP* system variables)

**Resulting Dimension** Displays a length dimensional tooltip that is updated as you move the grip.

**Length Change** Displays the change in length as you move the grip.

**Absolute Angle** Displays an angle dimensional tooltip that is updated as you move the grip.

**Angle Change** Displays the change in the angle as you move the grip.

**Arc Radius** Displays the radius of an arc, which is updated as you move the grip.

# Tooltip Appearance Dialog Box

## Quick Reference

Tools ➤ Drafting Settings
Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar and click Settings.
**dsettings** (or '**dsettings** for transparent use)

Controls the appearance of tooltips.

**NOTE** Use the *TOOLTIPMERGE* system variable to combine drafting tooltips into a single tooltip.

### Previews

Displays an example of the current tooltip appearance settings.

**Colors** Displays the Drawing Window Colors dialog box (page 983), where you can specify a color for drafting tooltips and their backgrounds in a specified context.

### Size

Specifies a size for tooltips. The default size is 0. Use the slider to make tooltips larger or smaller.

#### Transparency

Controls the transparency of tooltips. The lower the setting, the less transparent the tooltip. A value of 0 sets the tooltip to opaque.

#### Apply To

Specifies whether the settings apply to all drafting tooltips or only to Dynamic Input tooltips. (*DYNTOOLTIPS* system variable)

**Override OS Settings for All Drafting Tooltips** Applies the settings to all tooltips, overriding the settings in the operating system.

**Use Settings Only for Dynamic Input Tooltips** Applies the settings only to the drafting tooltips used in Dynamic Input.

# DSVIEWER

### Quick Reference

Opens the Aerial View window

View ➤ Aerial ViewAt the Command prompt, enter dsviewer.
**dsviewer**

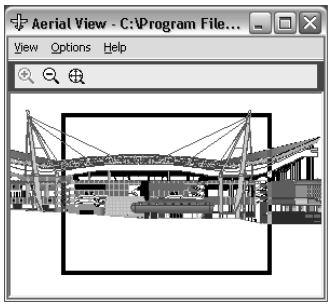
The Aerial View window (page 501) is displayed.

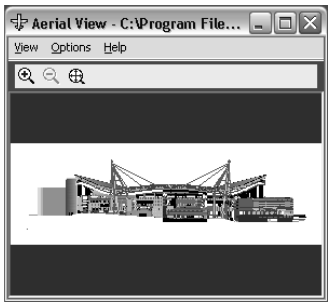
## Aerial View Window

### Quick Reference

View ➤ Aerial View
**dsviewer**

Displays the entire drawing; the current view is marked with a wide outline box.



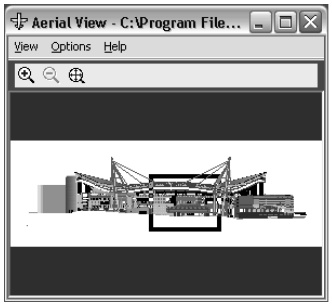


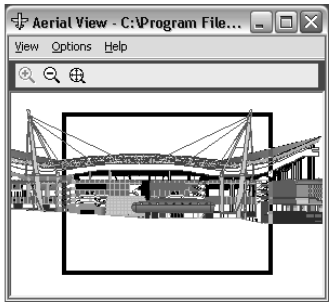

**View Menu (Aerial View Window)**

Changes the magnification of the Aerial View by zooming in and out of the drawing or by displaying the entire drawing in the Aerial View window.

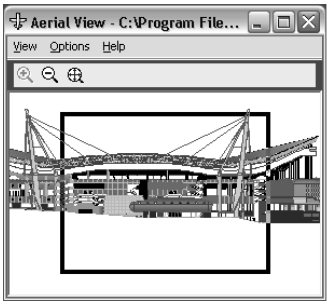
When the entire drawing is displayed in the Aerial View window, the Zoom Out menu option and button are unavailable. When the current view nearly fills the Aerial View window, the Zoom In menu option and button are unavailable. If both of these conditions exist at the same time, such as after using *ZOOM* Extents, both options are unavailable. All of the menu options are also available from a shortcut menu you can access by right-clicking in the Aerial View window.

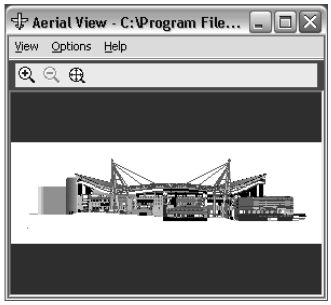
**Zoom In** Increases the magnification of the drawing in the Aerial View window by zooming in by a factor of 2, centered on the current view box.



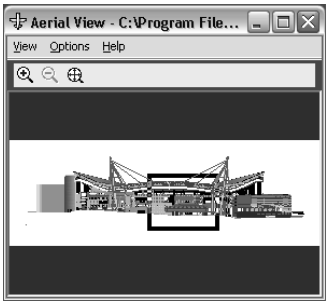


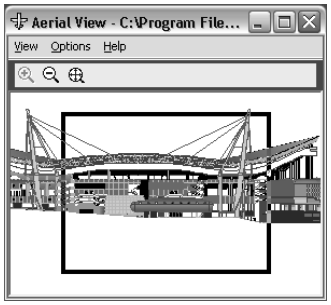

**Zoom Out** Decreases the magnification of the drawing in the Aerial View window by zooming out by a factor of 2, centered on the current view box.




**Global** Displays the entire drawing and the current view in the Aerial View window.




### Options Menu (Aerial View Window)

Provides toggles for automatic viewport display and dynamic updating of the drawing. All of the menu options are also available from a shortcut menu you can access by right-clicking in the Aerial View window.

**Auto Viewport** Displays the model space view of the current viewport automatically when multiple viewports are displayed. When Auto Viewport is off, the Aerial View window is not updated to match the current viewport.

**Dynamic Update** Updates the Aerial View window while you edit the drawing. When Dynamic Update is off, the Aerial View window is not updated until you click in the Aerial View window.

**Realtime Zoom** Updates the drawing area in real time when you zoom using the Aerial View window.

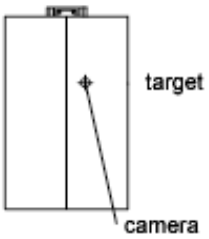
# DVIEW

## Quick Reference

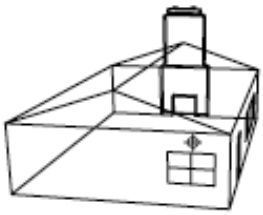
Defines parallel projection or perspective views by using a camera and target

**dview**
Select objects (page 504) or <use DVIEWBLOCK (page 504)>.
Enter option
[CAmera (page 505)/TArget (page 506)/Distance (page 508)/POints (page 509)/PAn (page 509)/Zoom (page 510)/TWist (page 511)/CLip (page 511)/Hide (page 512)/Off (page 512)/Undo (page 512)]: Specify a point (page 505) *with your pointing device, or enter an option*

**NOTE** Transparent *ZOOM*, *DSVIEWER*, *PAN*, and scroll bars are not available in DVIEW. When you define a perspective view, ZOOM, PAN, transparent ZOOM and PAN, DSVIEWER, and scroll bars are not available while that view is current.



plan view

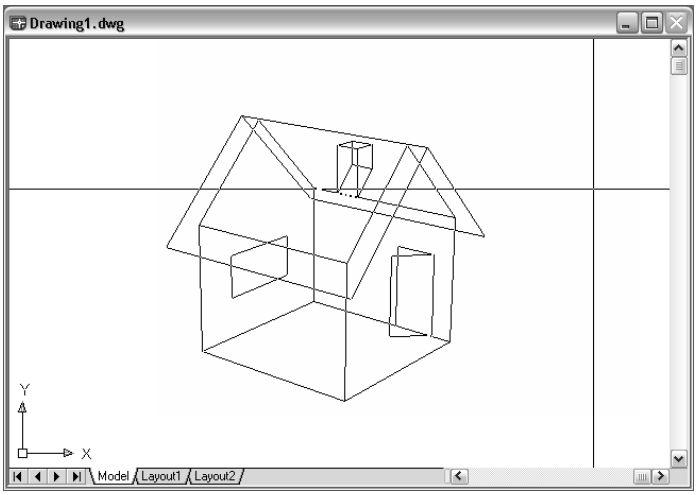

3D perspective view

### Object Selection

Specifies objects to use in the preview image as you change views. Selecting too many objects slows image dragging and updating.

### DVIEWBLOCK

If you press ENTER at the Select Objects prompt, DVIEWBLOCK displays a preview image. You can create your own DVIEWBLOCK block in a 1 unit by 1 unit by 1 unit area, with its origin at the lower-left corner. The following illustration shows an example of using the default DVIEWBLOCK to set the view (moving the graphics cursor adjusts the view).




### Point Specification

Rolls the view under the camera. The point you select with your pointing device is a start point for the dragging operation. Your viewing direction changes about the target point as you move the pointing device.

Enter direction and magnitude angles: *Enter angles between 0 degrees and 360 degrees, or specify a point with your pointing device*

Enter both angles, separated by a comma. The angles must be positive. The direction angle indicates the front of the view, and the magnitude angle determines how far the view rolls.

### Camera

Specifies a new camera position by rotating the camera about the target point. Two angles determine the amount of rotation.

Specify camera location, or enter angle in XY plane <*from* X *axis*>, or [Toggle (angle *current*)] <*current*>: *Specify an XYZ point, enter* **t**, *enter an angle, or press* ENTER

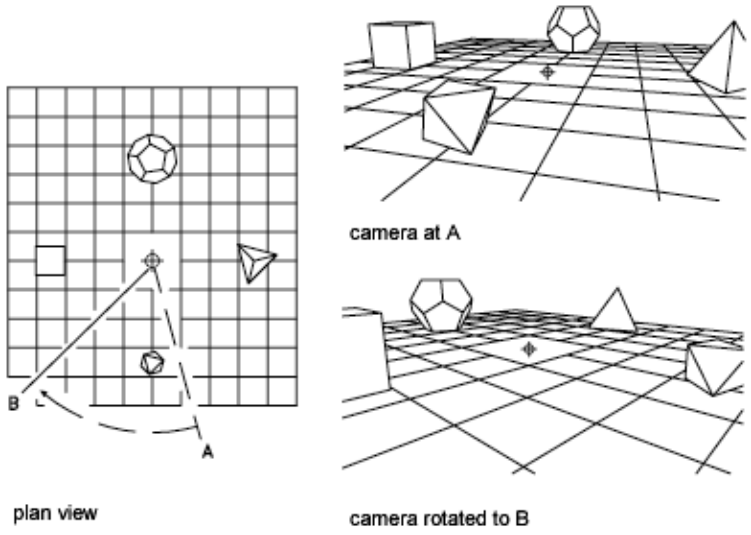
**Camera Location** Sets the camera's position based on the specified point.

**Enter Angle from the XY Plane** Sets the camera's position at an angle above or below the *XY* plane. An angle of 90 degrees looks down from above, and an angle of -90 looks up from below. A camera angle of 0 degrees places the camera parallel to the *XY* plane of the user coordinate system (UCS).

When you toggle the angle input mode or specify the angle from the *XY* plane, you are returned to the previous prompt.

**Toggle (Angle In)** Switches between two angle input modes. Entering an angle at the command prompt locks the cursor movement so you see only the positions available for that angle. Toggle unlocks the cursor movement for the angle, and you can use the cursor to rotate the camera.

**Enter Angle in XY Plane from X Axis** Sets the camera's position at an angle in the *XY* plane relative to the *X* axis of the current UCS. This angle measures from -180 to 180 degrees. A rotation angle of 0 degrees looks down the *X* axis of the UCS toward the origin.




The illustration shows the camera rotating to the left from its initial position, leaving its angle from the *XY* plane unchanged.

**Toggle (Angle From)** Switches between two angle input modes. Entering an angle at the command prompt locks the cursor movement so you see only the positions available for that angle. Toggle unlocks the cursor movement for the angle, and you can use the cursor to rotate the camera.

### Target

Specifies a new position for the target by rotating it around the camera. The effect is like turning your head to see different parts of the drawing from one vantage point. Two angles determine the amount of rotation.

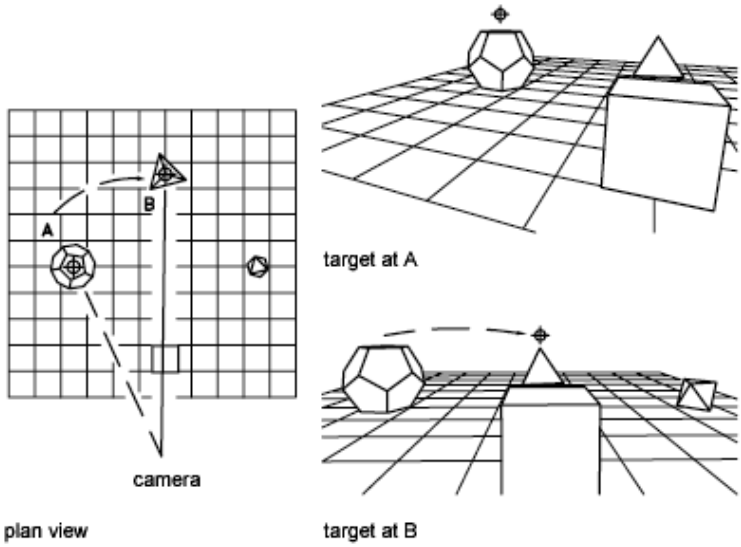
Specify camera location, or enter angle in XY plane <*from* X *axis*>, or [Toggle (angle *current*)] <*current*>: *Specify an XYZ point, enter* **t**, *enter an angle, or press* ENTER

**Enter Angle from the XY Plane** Sets the target's position at an angle above or below the *XY* plane. An angle of 90 degrees looks down from above, and an angle of -90 looks up from below. A target angle of 0 degrees means that the target is parallel to the *XY* plane of the UCS.

After you toggle the angle input mode or specify the angle from the *XY* plane, you are returned to the previous prompt.

**Toggle (Angle In)** Switches between two angle input modes. Entering an angle at the command prompt locks the cursor movement so you see only the positions available for that angle. Toggle unlocks the cursor movement for the angle, and you can use the cursor to rotate the target.

**Enter Angle in XY Plane from X Axis** Sets the target's position at an angle in the *XY* plane relative to the *X* axis of the current UCS. This angle measures from -180 to 180 degrees. A rotation angle of 0 degrees means you look down the *X* axis of the UCS toward the origin.




The illustration shows the effect of moving the target point from left to right, leaving its angle from the *XY* plane unchanged.

**Toggle (Angle From)** Switches between two angle input modes. Entering an angle at the command prompt locks the cursor movement so you see only

the positions available for that angle. Toggle unlocks the cursor movement for the angle, and you can use the cursor to rotate the target.

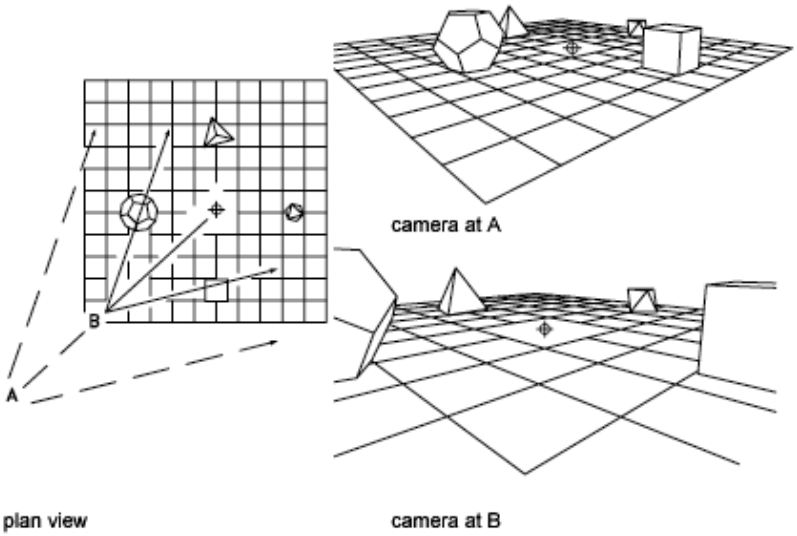
### Distance

Moves the camera in or out along the line of sight relative to the target. This option turns on perspective viewing, which causes objects farther from the camera to appear smaller than those closer to the camera. A special perspective icon replaces the coordinate system icon. You are prompted for the new camera-to-target distance.

Specify new camera-target distance <*current*>: *Enter a distance or press* ENTER

A slider bar along the top of the drawing area is labeled from 0x to 16x, with 1x representing the current distance. Moving the slider bar to the right increases the distance between camera and target. Moving it to the left decreases that distance. To turn off perspective viewing, click the Off option from the main DVIEW prompt.

If the target and camera points are close together, or if you specify a long-focal-length lens, you might see very little of your drawing when you specify a new distance. If you see little or none of your drawing, try the maximum scale value (16x) or enter a large distance. To magnify the drawing without turning perspective viewing on, use the Zoom option of DVIEW.




The illustration shows the effect of moving the camera along the line of sight relative to the target, where the field of view remains constant.

### Points

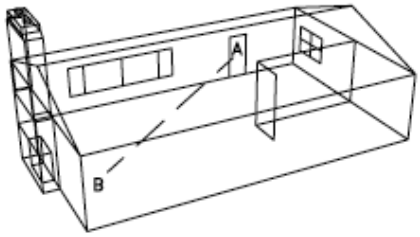
Locates the camera and target points using *X*,*Y*,*Z* coordinates. You can use *XYZ* point filters.

Specify target point <*current*>: *Specify a point or press* ENTER

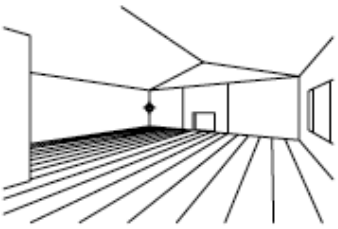
To help you define a new line of sight, a rubber-band line is drawn from the current camera position to the crosshairs. You are prompted for a new camera location.

Specify camera point <*current*>: *Specify a point, enter direction and magnitude angles, or press* ENTER

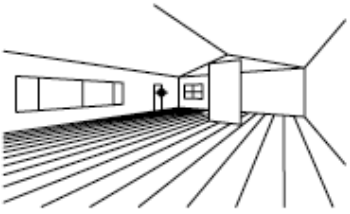
A rubber-band line connects the target point to the crosshairs to help you place the camera relative to the target. The illustration shows the change in view as you swap the camera and target points. Lens and distance settings are the same in each case.




camera location and target point



camera at A, target at B



camera at B, target at A

For information about entering direction and magnitude angles, see Point Specification.

### Pan

Shifts the image without changing the level of magnification.

Specify displacement base point: *Specify a point*
Specify second point: *Specify a point*

### Zoom

If perspective viewing is off, dynamically increases or decreases the apparent size of objects in the current viewport.

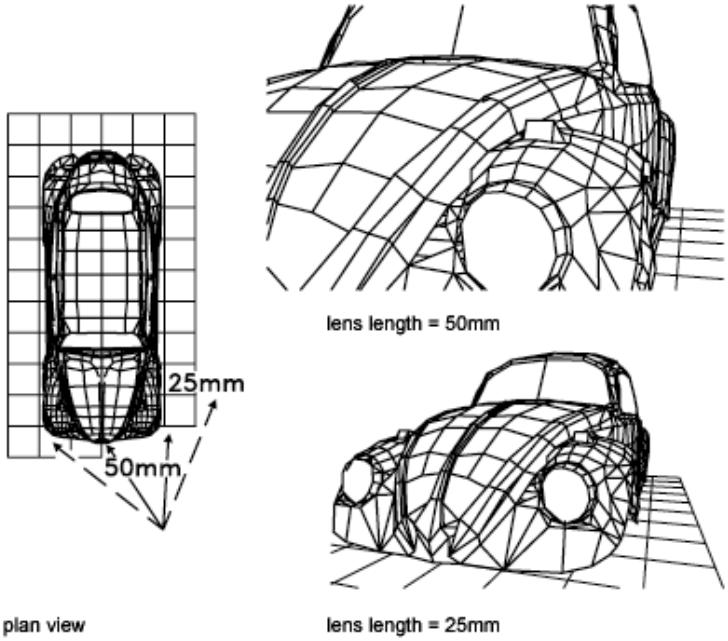
Specify zoom scale factor <*current*>: *Specify a scale or press* ENTER

A slider bar along the top of the drawing area is labeled from 0x to 16x, with 1x representing the current scale. Moving the slider bar to the right increases the scale. Moving it to the left decreases the scale.

If perspective viewing is on, Zoom adjusts the camera lens length, which changes the field of view and causes more or less of the drawing to be visible at a given camera and target distance. The default lens length is 50mm, simulating what you'd see with a 35mm camera and a 50mm lens. Increasing the lens length is similar to switching to a telephoto lens. Decreasing the lens length widens the field of view, as with a wide-angle lens.

Specify lens length <50.000mm>: *Specify a value or press* ENTER

A slider bar along the top of the drawing area is labeled from 0x to 16x, with 1x representing the current lens length. Moving the slider bar to the right increases the lens length. Moving it to the left decreases the lens length.



### Twist

Twists or tilts the view around the line of sight. The twist angle is measured counterclockwise, with 0 degrees to the right.

Specify view twist angle <*current*>: *Specify an angle or press* ENTER

### Clip

Clips the view, obscuring portions of the drawing that are behind or in front of the front clipping plane. The front and back clipping planes are invisible walls that you can position perpendicular to the line of sight between the camera and target.

Enter clipping option [Back/Front/Off] <Off>: *Enter an option or press* ENTER

### Back

Obscures objects located behind the back clipping plane.

Specify distance from target or [ON/OFF] <*current*>: *Specify a distance, enter an option, or press* ENTER
**Distance from Target** Positions the back clipping plane and turns on back clipping. A positive distance places the clipping plane between the target and the camera. A negative distance places it beyond the target. You can use the slider bar to drag the clipping plane.

**On** Turns on back clipping at the current clipping distance.

**Off** Turns off back clipping.

### Front

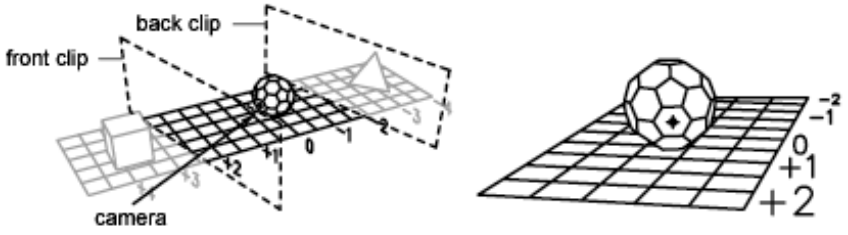
Obscures objects located between the camera and the front clipping plane.

Specify distance from target or [set to Eye (camera)/ON/OFF] <*current*>: *Specify a distance, enter* **e**, *or press* ENTER
**Distance from Target** Positions the front clipping plane and turns on front clipping. A positive distance places the clipping plane between the target and the camera. A negative distance places it beyond the target. You can use the slider bar to drag the clipping plane.

**Eye** Positions the front clipping plane at the camera.

**On** Turns on front clipping. This option is available only when perspective viewing is off.

**Off** Turns off front clipping. This option is available only when perspective viewing is off.




### Off

Turns off front and back clipping. If perspective viewing is on, front clipping remains on at the camera position.

### Hide

Suppresses hidden lines on the selected objects to aid in visualization. Circles, solids, traces, regions, wide polyline segments, 3D faces, polygon meshes, and the extruded edges of objects with nonzero thickness are considered to be opaque surfaces that hide objects.This hidden line suppression is quicker than that performed by *HIDE*, but it can't be plotted.

### Off

Turns off perspective viewing. The Distance option turns on perspective viewing.

### Undo

Reverses the effects of the last DVIEW action. You can undo multiple DVIEW operations.

# DWFADJUST

## Quick Reference

Allows adjustment of a DWF underlay at the command prompt

**dwfadjust**
Select DWF underlay: *Select one or more DWF underlays*
Enter DWF underlay option [Fade (page 513)/Contrast (page 513)/Monochrome (page 513)] <Fade>:

If you selected a single DWF underlay, the default values for Fade, Contrast, and Monochrome are the current property settings of the underlay selected. If you select multiple underlays, the default values for Fade, Contrast, Monochrome remain as they were set the last time the command was used.

To adjust colors in the DWF underlay, open the Properties palette for the DWF underlay. For more information, see Adjust DWF Underlay Contrast, Fade, Monochrome, and Colors for Background.

### Fade

Controls the fade effect of the underlay. Values range from 0 through 80. The greater the value, the lighter the linework in the underlay appears. Works indirectly with the contrast effect; a higher contrast value blends the underlay into the background when fade is set to a higher value. Default=25.

Enter fade value (0-80): *Enter a value*

### Contrast

Controls the contrast, and indirectly the fading effect, of the underlay. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color. Default=75.

Enter contrast value (0-100): *Enter a value*

### Monochrome

This Yes/No toggle controls the color saturation of all linework while maintaining the luminance. When turned on, the linework appears in varying shades of gray starting at black if the background color luminance is 50% or more. If the background color luminance is less than 50%, then the colors are

inverted, with the darkest linework displaying in white, and the lightest linework displaying in black.

Monochrome? [Yes/No]: *Select* **yes** *or* **no** *and then press ENTER*

# DWFATTACH

## Quick Reference

Attaches a DWF underlay to the current drawing


Insert

Insert ➤ DWF UnderlayAt the Command prompt, enter dwfattach.
**dwfattach**

The Select DWF File dialog box (a standard file selection dialog box (page 931)) is displayed. Once you select a DWF file, the Attach DWF Underlay dialog box (page 514) is displayed.

**NOTE** If you plan to access DWF files from the Vault client server, the Vault file open dialog box supersedes the Select DWF File dialog box. Access to the Vault client server is only available to Autodesk Subscription customers.

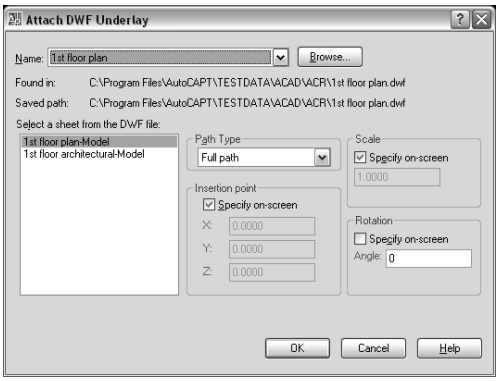
# Attach DWF Underlay Dialog Box

## Quick Reference


Insert

Insert ➤ DWF UnderlayAt the Command prompt, enter dwfattach.
**dwfattach**

Names, locates and defines the insertion point, scale and rotation of attached DWF underlays.




### Name

Identifies the DWF file you have selected to attach, either from the Select DWF File dialog box (an unattached DWF file) or from the list of previously attached DWF underlays. To add another instance of a DWF underlay that is already attached, select the DWF name from the list.

---

**NOTE** The Name field is disabled when you're attaching a DWF file that is stored on the Vault client server. This information is automatically entered by the Vault. Access to the Vault client server is only available to Autodesk Subscription customers.

---

**Browse** Opens the Select DWF File dialog box (a standard file selection dialog box (page 931)). If Views ➤ Preview is selected, a preview of the selected file is displayed.

---

**NOTE** The Browse button is hidden when you're attaching a DWF file that is stored on the Vault client server.

---

**Found In** Displays the path where the DWF file was located.

**Saved Path** Displays the path that is saved with the drawing when the DWF file is attached. The path is dependent upon the Path Type setting.

### Select a Sheet

Displays all of the sheets that are found in the DWF file. If the DWF file only contains a single sheet, that sheet is listed.

If the DWF file contains multiple sheets, only a single sheet can be selected for attachment. The first sheet in the list is selected by default.

**NOTE** 3D sheets are not listed and if a DWF file does not contain a usable 2D sheet, a warning message is displayed.

### Path Type

Specifies one of three types of folder path information to save with an attached DWF underlay: a full path, a relative path, and no path. For a complete description of each option, see "Set Paths to Referenced Drawings" in the *User's Guide*.

**NOTE** The Path Type group is disabled when you're attaching a DWF file that is stored on the Vault client server. This information is automatically entered by the Vault.

**Full Path** Specifies the full (absolute) path to the DWF file.

**Relative Path** Specifies a relative path to the DWF file.

**No Path** Specifies only the DWF file name. The DWF file should be located in the folder with the current drawing file.

### Insertion Point

Specifies the insertion point for the selected DWF file. Specify On-Screen is the default. The default insertion point is 0,0,0.

**Specify On-Screen** Directs input at the command prompt or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.

**X** Sets the *X* coordinate value.

**Y** Sets the *Y* coordinate value.

**Z** Sets the *Z* coordinate value.

### Scale

Specifies the scale factor of the selected DWF underlay.

Specify On-Screen directs input at the command prompt or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.

If INSUNITS (page 1727) is set to "unitless" or if the underlay does not contain resolution information, the scale factor becomes the underlay width in AutoCAD units. If INSUNITS has a value such as millimeters, centimeters,

inches, or feet, and the underlay has resolution information, the scale factor is applied after the true width of the underlay in AutoCAD units is determined.

#### Rotation

Specifies the rotation angle of the selected DWF underlay.

If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value at the command prompt. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

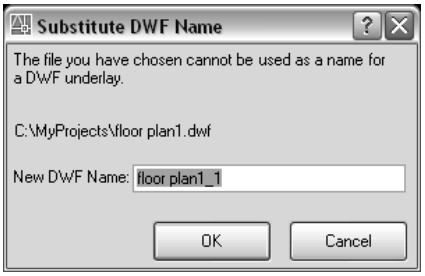
## Substitute DWF Name Dialog Box

### Quick Reference

Insert 

Insert ➤ DWF UnderlayAt the Command prompt, enter dwfattach.
**dwfattach**

The names of DWF references you attach must be unique. Attempts to attach two DWF references that have the same name results in displaying the Substitute DWF Name dialog box.




#### New DWF Name

Enter a unique name for the DWF underlay you are attempting to attach.

Once a new name is supplied, you can continue with the attachment process.

**NOTE** This only changes the name of the DWF reference when it is attached. It does not affect the name of the DWF file.

# -DWFATTACH

## Quick Reference

If you enter **-dwfattach** at the command prompt, the following DWFATTACH command prompts are displayed.

Path to DWF file to attach: *Enter the path and filename of the DWF file to be attached*
Enter name of sheet or [? (page 518)]<First valid sheet name in the drawing>: *Specify a sheet name or press ENTER to accept the sheet name that is listed*
Specify insertion point: *Pick an insertion point*
Specify scale factor or [Unit] <1.0000>: *Specify a scale factor*
Specify rotation <0>: *Specify a rotation angle*

### ?—List Sheets

Lists the valid sheets in the current drawing.

Enter sheet(s) to list <*>: *Enter a name, a partial name with wild-card characters, or press* ENTER *to list all sheets*

After the sheets are listed, the previous prompt returns.

# DWFCLIP

## Quick Reference

Uses clipping boundaries to define a subregion of a DWF underlay

Select a DWF underlay to clip, right-click in the drawing area, and choose DWF Clip.
**dwfclip**
Select DWF to clip: *Select an edge of a DWF underlay*
Enter DWF clipping option [ON (page 519)/OFF (page 519)/Delete (page 519)/New boundary (page 519)] <New boundary>: *Enter an option or press* ENTER

The boundary you specify must be in a plane parallel to the DWF underlay.

### On

Turns on clipping and displays the DWF underlay clipped to the previously defined boundary.

### Off

Turns off clipping and displays the entire DWF underlay and frame.

If you reclip the DWF underlay while clipping is turned off, clipping is automatically turned back on. You are prompted to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.

### Delete

Removes a predefined clipping boundary and redisplays the full original underlay.

### New Boundary

Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the DWF underlay boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the point is interpreted as the first corner of a rectangle.

Enter clipping type [Polygonal/Rectangular] <Rectangular>: *Enter* **p** *or press* ENTER

**Polygonal** Uses specified points to define a polygonal boundary.

Specify first point: *Specify a point*

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Close/Undo]: *Specify a point, or enter* **c** *or* **u**

You must specify at least three points to define a polygon.

If the DWF underlay already has a clipping boundary defined, the following prompt is displayed:

Delete old boundary? [No/Yes] <Yes>: *Enter* **n** *or press* ENTER

If you choose Yes, the entire DWF underlay is redrawn and the command continues; if you choose No, the command ends.

**Rectangular** Specifies a rectangular boundary by its opposite corners. The rectangle is always drawn parallel to the edges of the DWF underlay.

Specify first corner point: *Specify a point*
Specify opposite corner point: *Specify a point*

# DWFLAYERS

## Quick Reference

Controls the display of layers in a DWF underlay

Select a DWF underlay, right-click in the drawing area, and choose DWF Layers.
**dwflayers**
Select DWF underlay: *Select a DWF underlay in which to manage the layer display*

After selecting the DWF underlay, the DWF Layers dialog box (page 520) is displayed.

# DWF Layers Dialog Box

## Quick Reference

Select a DWF underlay, right-click in the drawing area, and choose DWF Layers.
**dwflayers**

Displays a list of the layers in the DWF underlay and their display status.

---

**NOTE** It is possible for a DWF file to be published without any layer information. If an attached DWF underlay does not include at least one layer, the DWF Layers dialog box cannot be displayed.

---

**On** Turns the selected layers on and off.

**Name** Displays the name of the layer.

To turn a selected layer on or off, click its light bulb icon. You can select several layers by pressing the CTRL or SHIFT keys while clicking layer names. To turn several selected layers on and off, right-click and choose Layer(s) On or Layer(s) Off.

# DWGPROPS

## Quick Reference

Sets and displays the properties of the current drawing

File ➤ Drawing PropertiesAt the Command prompt, enter dwgprops.
**dwgprops**

The Drawing Properties dialog box (page 521) is displayed.

# Drawing Properties Dialog Box

## Quick Reference

File ➤ Drawing Properties
**dwgprops**

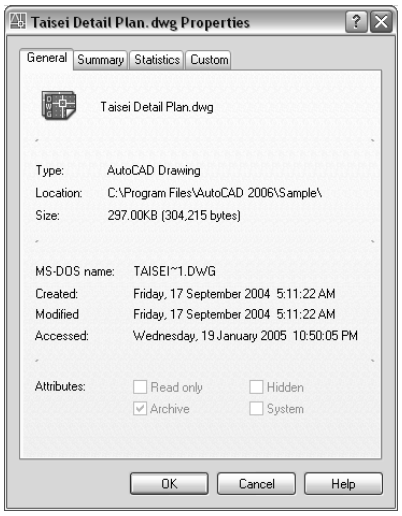
Displays read-only statistics or general information about your drawing, assigns summary properties, and assigns names and values to custom properties. These custom properties can help you identify your drawing.

Document properties are displayed in the list of field names in the Field dialog box (page 578).

- General (page 521)
- Summary (page 523)
- Statistics (page 524)
- Custom (page 525)

### General Tab (Drawing Properties Dialog Box)

Displays read-only information about the drawing file. This data is derived from the operating system.




### File Name

Shows the file icon and the file name.

### File Type, Location, Size

Shows the file type, the file location, and the size of the file.

### MS-DOS Name, Created, Modified, Accessed

Shows MS-DOS name, when the file was created, and the date and time it was last modified and last accessed.

### Attributes

Shows system-level file attributes. These values can be modified in Windows Explorer.

**Read-Only** Indicates that the file is read-only; it cannot be changed or deleted accidentally.

**Archive** Indicates that this file should be archived. This setting is used to determine which files should be backed up.

**Hidden** Indicates that the file is hidden; you cannot see or use it unless you know its name.

**System** Indicates that the file is a system file. A drawing cannot have the System attribute set.

### Summary Tab (Drawing Properties Dialog Box)

Displays properties such as author, title, and subject that are predefined. For example, you can add keywords to all your drawing files and then use DesignCenter™ to search for all drawing files with a particular keyword.


**Title** Specifies the title you want to use when searching for this drawing. The title can be different from the drawing file name.

**Subject** Specifies the subject of the drawing. You can use the subject name to group drawings that have the same subject.

**Author** Specifies the author of the drawing. The author name can only be entered or changed by the user. To change the author, delete the existing name and enter a new one.

**Keywords** Specifies the keywords you want to use to locate the drawing.

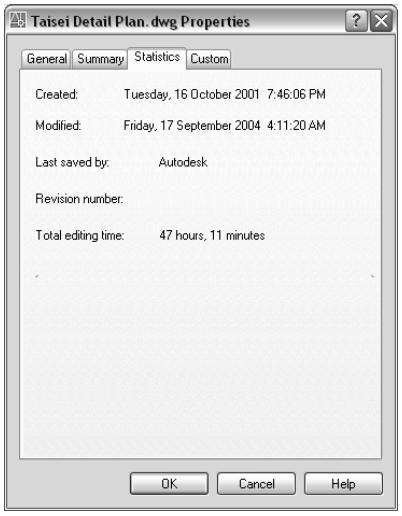
**Comments** Provides a space to add comments to the drawing.

**Hyperlink Base** Specifies the base address that is used for all relative links inserted within the drawing. You can specify an Internet location, for example, *http://www.autodesk.com*, or a path to a folder on a network drive.

### Statistics Tab (Drawing Properties Dialog Box)

Displays data such as the dates the drawing was created and last modified. These file properties are automatically maintained for you and can help you search for drawings created or modified during a specific period.

**NOTE** If the drawing was last saved using an application other than Autodesk® software, a warning message is displayed. Display of the warning message is controlled by the *DWGCHECK* system variable.




**Created** Displays the date and time the drawing was created. This value is stored in the *TDCREATE* system variable.

**Modified** Displays the date and time the drawing was last modified. This value is stored in the *TDUPDATE* system variable.

**Last Saved By** Displays the name of the last person who modified the file. The Last Saved By name is stored in the *LOGINNAME* system variable.

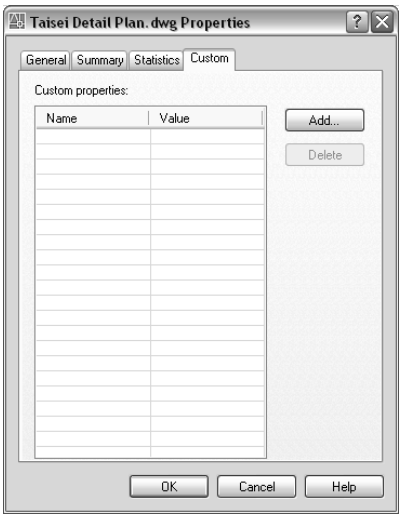
**Revision Number** Displays the revision number. This value is not accessible. To track revisions, create a custom property on the Custom tab.

**Total Editing Time** Displays the total amount of editing time in the drawing. This value is stored in the *TDINDWG* system variable.

### Custom Tab (Drawing Properties Dialog Box)

Assigns custom properties to the drawing. For example, you could create a custom property called Project and assign the actual project name as the value. To assign the same custom properties to a group of drawings, create the custom properties in a drawing template file.

Custom properties are displayed in the list of field names in the Field dialog box. You also have access to the properties data using programming interfaces, such as AutoLISP®.




**Custom Properties** Lists names and values of custom properties for the current drawing.

**Add** Displays the Add Custom Property dialog box (page 526), where you can enter a name and a value for the new custom property. The name must be unique. The value can be left blank.

**Delete** Deletes the custom property that is selected in the list.

---

**NOTE** If you delete a custom property that is used in a field, the field displays #### the next time it is updated.

---

## Add Custom Property Dialog Box

### Quick Reference

File ➤ Drawing Properties
**dwgprops**

Adds a custom property to the drawing file.

**Custom Property Name** Specifies a unique name for the custom property. The name can contain up to 255 characters. The following characters are not permitted: asterisk (*), equal sign (=), less-than and greater-than signs (< >), slash (/), backslash (\), quotation marks ("), reverse quote (`), pipe sign (|), colon (:), and semicolon (;).

**Value** Specifies a value for the property. The value can be left blank.

# DXBIN

### Quick Reference

Imports specially coded binary files

Insert ➤ Drawing Exchange BinaryAt the Command prompt, enter dxbin.
**dxbin**

The Select DXB File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the name of the file to import.

# E Commands

# 6

**In this chapter**

- EATTEDIT
- EATTEXT
- EDGE
- EDGESURF
- ELEV
- ELLIPSE
- ERASE
- ETRANSMIT
- EXPLODE
- EXPORT
- EXPORTTOAUTOCAD
- EXTEND
- EXTERNALREFERENCES
- EXTERNALREFERENCESCLOSE
- EXTRUDE

# EATTEDIT

## Quick Reference

Edits attributes in a block reference

Modify II 
Modify ➤ Object ➤ Attribute ➤ SingleAt the Command prompt, enter eattedit.
**eattedit**
Select a block:

You are prompted to select a block in the drawing area. After you select a block with attributes, the Enhanced Attribute Editor (page 528) is displayed.

If the block you select does not contain attributes, or if you select something that is not a block, an error message isblock displayed, and you are prompted to select another block.

# Enhanced Attribute Editor

## Quick Reference

Modify II
Modify ➤ Object ➤ Attribute ➤ SingleAt the Command prompt, enter eattedit.
**eattedit**

Lists the attributes in a selected block instance and displays the properties of each attribute. You can change the attribute properties and values.

**Block** Name of the block whose attributes you are editing.

**Tag** Tag identifying the attribute.

**Select Block** Temporarily closes the dialog box while you select a block with your pointing device.

If you modify attributes of a block and then select a new block before you save the attribute changes you made, you are prompted to save the changes before selecting another block.

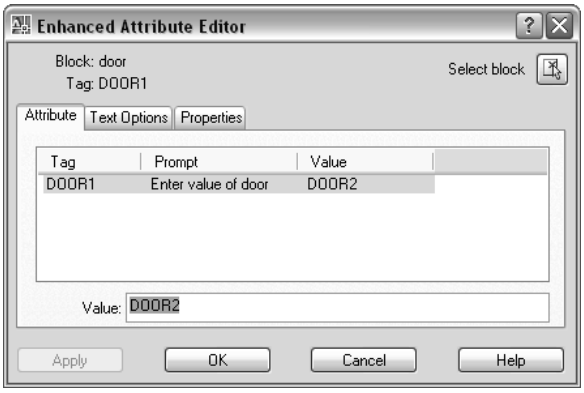
**Apply** Updates the drawing with the attribute changes you have made, and leaves the Enhanced Attribute Editor open.

The Enhanced Attribute Editor contains the following tabs:

- Attribute (page 529)
- Text Options (page 529)
- Properties (page 530)

### Attribute Tab (Enhanced Attribute Editor)

Displays the tag, prompt, and value assigned to each attribute. You can change only the attribute value.




**List** Lists the attributes in the selected block instance and displays the tag, prompt, and value for each attribute.

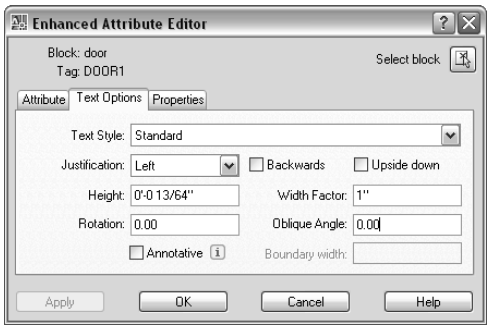
**Value** Assigns a new value to the selected attribute.

Multiple-line attributes include a button with an ellipsis. Click to open the In-Place Text Editor (page 873) with the Text Formatting toolbar and the ruler. Depending on the setting of the *ATTIPE* system variable, the Text Formatting toolbar displayed is either the abbreviated version, or the full version

To use a field as the value, right-click and click Insert Field on the shortcut menu to display the Field dialog box (page 578).

### Text Options Tab (Enhanced Attribute Editor)

Sets the properties that define the way an attribute's text is displayed in the drawing. Change the color of attribute text on the Properties tab.




**Text Style** Specifies the text style for the attribute text. Default values for this text style are assigned to the text properties displayed in this dialog box.

**Justification** Specifies how the attribute text is justified (left-, center-, or right-justified).

**Height** Specifies the height of the attribute text.

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

**Annotative** Specifies that the attribute is . Click the information icon to learn more about annotative objects.

**Backwards** Specifies whether or not the attribute text is displayed backwards. Not available for multiple-line attributes.

**Upside down** Specifies whether or not the attribute text is displayed upside down. Not available for multiple-line attributes.

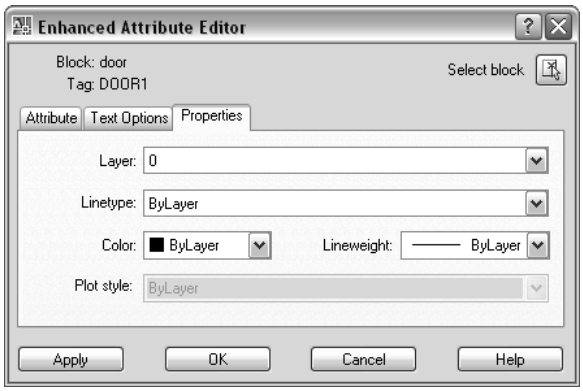
**Width Factor** Sets the character spacing for the attribute text. Entering a value less than **1.0** condenses the text. Entering a value greater than **1.0** expands it.

**Oblique Angle** Specifies the angle that the attribute text is slanted away from its vertical axis. Not available for multiple-line attributes.

**Boundary Width** Specifies the maximum length of the lines of text in a multiple-line attribute before wrapping to the next line. A value of 0.000 means that there is no restriction on the length of a line of text. Not available for single-line attributes.

### Properties Tab (Enhanced Attribute Editor)

Defines the layer that the attribute is on and the lineweight, linetype, and color for the attribute text. If the drawing uses plot styles, you can assign a plot style to the attribute using the Properties tab.




**Layer** Specifies the layer that the attribute is on.

**Linetype** Specifies the linetype of the attribute.

**Color** Specifies the color of the attribute.

**Plot Style** Specifies the plot style of the attribute.

If the current drawing uses color-dependent plot styles, the Plot Style list is not available.

**Lineweight** Specifies the lineweight of the attribute.

Changes you make to this option are not displayed if the LWDISPLAY system variable is off.

# EATTEXT

## Quick Reference

Exports property data from objects, block attribute information, and drawing information to a table or to an external file

Modify II

Tools ➤ Data ExtractionAt the Command prompt, enter dataextraction.
**eattext**

This command no longer displays the Attribute Extraction wizard and has been replaced by the Data Extraction wizard. (page 301)

If you enter **-eattext** at the command prompt, options are displayed at the command prompt. (page 532)

# -EATTEXT

## Quick Reference

If you enter **-eattext** at the command prompt, the following DATAEXTRACTION command prompts are displayed.

Extracts data as specified in an existing attribute extraction template (BLK) file created with the Attribute Extraction wizard in AutoCAD 2006 or data extraction (DXE) file created in AutoCAD 2007.

Enter the template file path for the extraction: type: *Specify the path and file name for the attribute extraction template (BLK) or data extractino (DXE) file that describes how to extract the information*

Subsequent prompts depend on the information set up in the template file. If the template specifies extracting data to an external file, the following prompts are displayed:

Enter the output filetype [Csv/Txt/Xls/Mdb] <Csv>: *Enter* **c** *for comma-separated (CSV),* **t** *for tab-separated (TXT),* **x** *for Microsoft Excel (XLS), or* **m** *for Microsoft Access (MDB)*
Enter output filepath: *Specify the names of the path and file where the data will be extracted*

---

**NOTE** The maximum number of columns that can be exported to an XLS and MDB file is 255.

---

If the template specifies a table, the following prompt is displayed:

Specify insertion point:

# EDGE

## Quick Reference

Changes the visibility of three-dimensional face edges

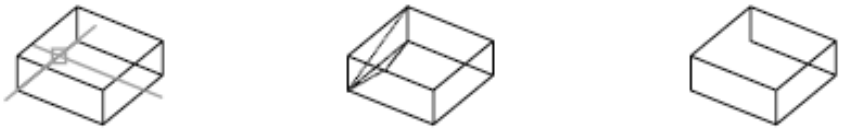
Draw ➤ Modeling ➤ Meshes ➤ EdgeAt the Command prompt, enter edge.
**edge**
Specify edge (page 533) of 3dface to toggle visibility or [Display (page 533)]: *Select an edge or enter* **d**

### Edge

Controls the visibility of the edges you select.

Specify edge of 3d face to toggle visibility or [Display]:

The prompt is repeated until you press ENTER.

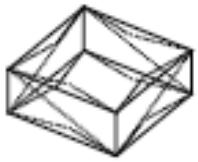


If the edges of one or more 3D faces are collinear, the program alters the visibility of each collinear edge.

### Display

Selects invisible edges of 3D faces so that you can redisplay them.

Enter selection method for display of hidden edges [Select/All] <All>: *Enter an option or press* ENTER



**All** Selects the hidden edges of all 3D faces in the drawing and displays them.

If you want to make the edges of the 3D faces visible once again, use the Edge option. You must select each edge with your pointing device to display it. AutoSnap™ markers and Snaptips are automatically displayed, indicating the apparent snap locations on each invisible edge.

This prompt continues until you press ENTER.

**Select** Selects hidden edges of a partially visible 3D face and displays them.

Select objects:

If you want to make the edges of the 3D faces visible once again, use the Edge option. You must select each edge with your pointing device to display it. AutoSnap markers and Snaptips are automatically displayed, indicating the apparent snap locations on each invisible edge.

This prompt continues until you press ENTER.

# EDGESURF

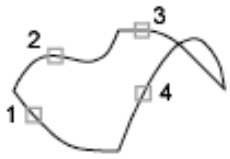
## Quick Reference

Creates a three-dimensional polygon mesh

Draw ➤ Modeling ➤ Meshes ➤ Edge MeshAt the Command prompt, enter edgesurf.

**edgesurf**

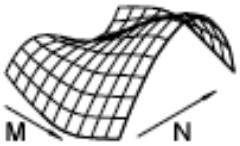
Current wire frame density: SURFTAB1=*current* SURFTAB2=*current*




Select object 1 for surface edge:
Select object 2 for surface edge:
Select object 3 for surface edge:
Select object 4 for surface edge:

You must select the four adjoining edges that define the mesh patch. The edges can be lines, arcs, splines, or open 2D or 3D polylines. The edges must touch at their endpoints to form a topologically rectangular closed path.

You can select the four edges in any order. The first edge (*SURFTAB1*) determines the *M* direction of the generated mesh, which extends from the endpoint closest to the selection point to the other end. The two edges that touch the first edge form the *N* edges (*SURFTAB2*) of the mesh.



# ELEV

## Quick Reference

Sets elevation and extrusion thickness of new objects

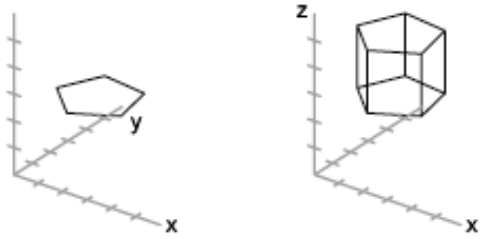
**elev** (or '**elev** for transparent use)
Specify new default elevation <*current*>: *Specify a distance or press* ENTER

The current elevation is the default *Z* value for new objects when you specify only *X* and *Y* values for a 3D point.

The elevation setting is the same for all viewports regardless of their user coordinate systems (UCSs). New objects are created at the specified *Z* value relative to the current UCS in the viewport.

Specify new default thickness <*current*>: *Specify a distance or press* ENTER

The thickness sets the distance to which a 2D object is extruded above or below its elevation. A positive value is extruded along the positive *Z* axis; a negative value is extruded along the negative *Z* axis.




with elevation  with thickness

ELEV controls only new objects; it does not affect existing objects. The elevation is reset to 0.0 whenever you change the coordinate system to the world coordinate system (WCS).

# ELLIPSE

## Quick Reference

Creates an ellipse or an elliptical arc

Draw 

Draw ➤ EllipseDoes not exist in the menus.

**ellipse**

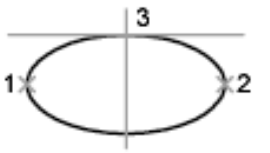
Specify axis endpoint (page 536) of ellipse or [Arc (page 537)/Center (page 539)/Isocircle (page 539)]: *Specify a point or enter an option*

The Isocircle option is available only when you set the Style option of *SNAP* to Isometric.

### Axis Endpoint

Defines the first axis by its two endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Specify other endpoint of axis: *Specify a point (2)*
Specify distance to other axis or [Rotation]: *Specify a distance by entering a value or locating a point (3), or enter* **r**
**Distance to Other Axis** Defines the second axis using the distance from the midpoint of the first axis to the endpoint of the second axis (3).

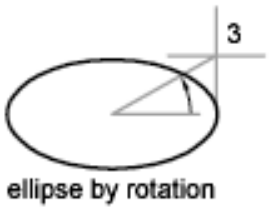



ellipse by axis endpoint

**Rotation** Creates the ellipse by appearing to rotate a circle about the first axis.

Specify rotation around major axis: *Specify a point (3), or enter an positive angle value less than* **90**

Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circular ellipse.




### Arc

Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Specify axis endpoint of elliptical arc or [Center]: *Specify a point or enter* **c**

### Axis Endpoint

Defines the start point of the first axis.

Specify other endpoint of axis:
Specify distance to other axis or [Rotation]: *Specify a distance or enter* **r**

The descriptions of the Distance to Other Axis and Rotation options match those of the corresponding options under Center.

### Center

Creates the elliptical arc using a center point you specify.

Specify center of elliptical arc:
Specify endpoint of axis:
Specify distance to other axis or [Rotation]: *Specify a distance or enter* **r**

### Distance to Other Axis

Defines the second axis as the distance from the center of the elliptical arc, or midpoint of the first axis, to the point you specify.

Specify start angle or [Parameter]: *Specify a point (1), enter a value, or enter* **p**

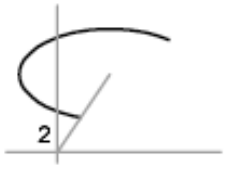
The descriptions of the Start Angle and Parameter options match those of the corresponding options under Rotation.


**Rotation**

Defines the major to minor axis ratio of the ellipse by rotating a circle about the first axis. The higher the value (from 0 through 89.4 degrees), the greater the ratio of minor to major axis. Entering **0** defines a circle.

Specify rotation around major axis: *Specify a rotation angle*
Specify start angle or [Parameter]: *Specify an angle or enter* **p**
**Start Angle** Defines the first endpoint of the elliptical arc. The Start Angle option toggles from Parameter mode to Angle mode. The mode controls how the ellipse is calculated.

Specify end angle or [Parameter/Included angle]: *Specify a point (2), enter a value, or enter an option*




**Parameter** Requires the same input as Start Angle, but creates the elliptical arc using the following parametric vector equation:

p(u) = c + a* cos(u) + b* sin(u)

where *c* is the center of the ellipse and *a* and *b* are its major and minor axes, respectively.

Specify start parameter or [Angle]: *Specify a point, enter a value, or enter* **a**

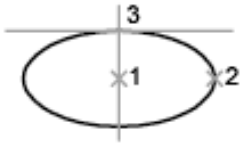
Specify end parameter or [Angle/Included angle]: *Specify a point, enter a value, or enter an option*

- *End Parameter:* Defines the end angle of the elliptical arc by using a parametric vector equation. The Start Parameter option toggles from Angle mode to Parameter mode. The mode controls how the ellipse is calculated.
- *Angle:* Defines the end angle of the elliptical arc. The Angle option toggles from Parameter mode to Angle mode. The mode controls how the ellipse is calculated.
- *Included Angle:* Defines an included angle beginning at the start angle.

### Center

Creates the ellipse by a center point you specify.

Specify center of ellipse: *Specify a point (1)*
Specify endpoint of axis: *Specify a point (2)*
Specify distance to other axis or [Rotation]: *Specify a distance by entering a value or locating a point (3), or enter* **r**




**Distance to Other Axis** Defines the second axis as the distance from the center of the ellipse, or midpoint of the first axis, to the point you specify.

**Rotation** Creates the ellipse by appearing to rotate a circle about the first axis.

Specify rotation around major axis: *Specify a point, or enter an angle value between 0 and 89.4*

Specify start angle or [Parameter]: *Specify an angle or enter* **p**

Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circle.

### Isocircle

Creates an isometric circle in the current isometric drawing plane.

---

**NOTE** The Isocircle option is available only when you set the Style option of *SNAP* to Isometric.

---

Specify center of isocircle:
Specify radius of isocircle or [Diameter]: *Specify a distance or enter* **d**

**Radius** Creates a circle using a radius you specify.

**Diameter** Creates a circle using a diameter you specify.

Specify diameter of isocircle: *Specify a distance*

# ERASE

## Quick Reference

Removes objects from a drawing

Modify 

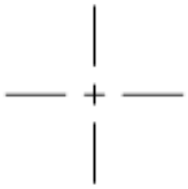
Modify ➤ EraseAt the Command prompt, enter erase.

Select the objects to erase, right-click in the drawing area, and click Erase.

**erase**

Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*

The objects are removed from the drawing.


# ETRANSMIT

## Quick Reference

Packages a set of files for Internet transmission

File ➤ eTransmit

**etransmit**

The Create Transmittal dialog box (page 541) is displayed.

If you enter **-etransmit** at the command prompt, options are displayed at the command prompt (page 552).

# Create Transmittal Dialog Box

## Quick Reference

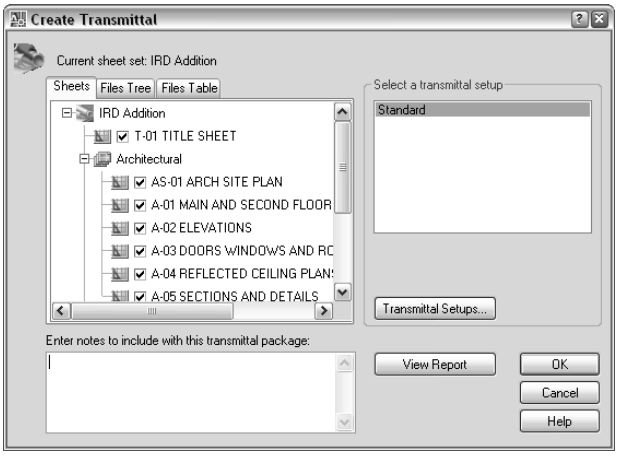
File ➤ eTransmit
**etransmit**

Packages a set of files for Internet transmission.

All files to be included in the transmittal package are indicated by a check mark next to the file name. Right-click in the file display area to display a shortcut menu with several options.

### Sheets Tab

Lists the sheets to be included in the transmittal package in a hierarchy based on sheet subsets. On this tab, you can create a transmittal package from a sheet set, sheet subset, or sheet. A sheet set must be open in the Sheet Set Manager, and eTransmit must be selected from the shortcut menu that is displayed when a sheet set, sheet subset, or sheet node is right-clicked.



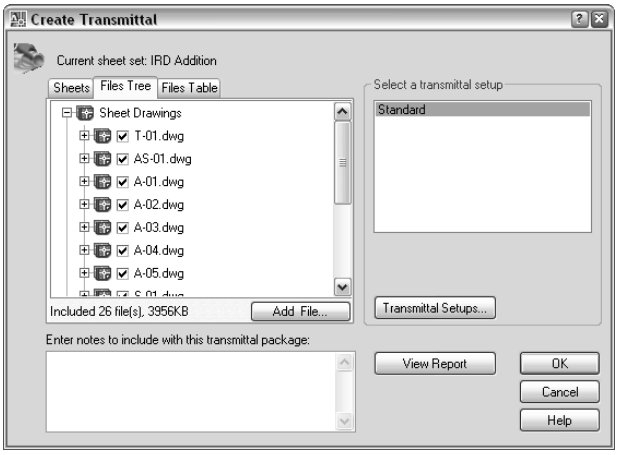

**NOTE** If a sheet in the list is unavailable, the sheet is referenced (as an xref) by another sheet in the transmittal package, and the unavailable sheet is automatically included in the transmittal package.

### Files Tree Tab

Lists the files to be included in the transmittal package in a hierarchical tree format. By default, all files associated with the current drawing (such as related xrefs, plot styles, and fonts) are listed. You can add files to the transmittal package or remove existing files. Related files that are referenced by URLs are not included in the transmittal package.

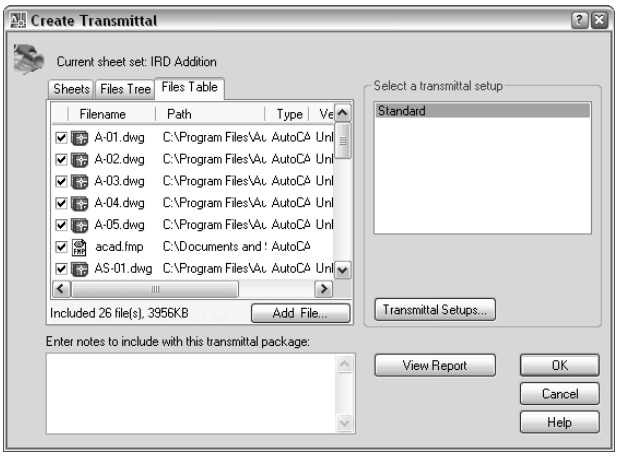
The drawings to be transmitted are listed under the following categories:

- **Sheet Drawings.** Lists the drawing files associated with the sheet set.
- **Sheet Set Files.** Lists the support files associated with the sheet set.
- **Current Drawing.** Lists the files associated with the current drawing.
- **User Added Files.** Lists the files that have been added manually with the Add File option.




### Files Table Tab

Displays the files to be included in the transmittal package in a table format. By default, all files associated with the current drawing (such as related xrefs, plot styles, and fonts) are listed. You can add files to the transmittal package or remove existing files. Related files that are referenced by URLs are not included in the transmittal package.




### Add File

Opens a standard file selection dialog box (page 931), in which you can select an additional file to include in the transmittal package. This button is available on both the Files Tree tab and the Files Table tab.

### Enter Notes to Be Included with This Transmittal Package

Provides a space where you can enter notes related to a transmittal package. The notes are included in the transmittal report. You can specify a template of default notes to be included with all your transmittal packages by creating an ASCII text file called *etransmit.txt.* This file must be saved to a location specified by the Support File Search Path option on the Files tab (page 948) in the Options dialog box.

### Select a Transmittal Setup

Lists previously saved transmittal setups. The default transmittal setup is named STANDARD. Click to select a different transmittal setup. To create a new transmittal setup or to modify an existing one in the list, click Transmittal Setups. Right-click to display a shortcut menu with several options.

### Transmittal Setups

Displays the Transmittal Setups dialog box (page 544), in which you can create, modify, and delete transmittal setups.

### View Report

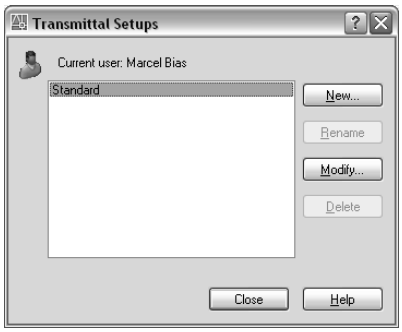
Displays report information that is included with the transmittal package. Includes any transmittal notes that you entered and distribution notes automatically generated that detail what steps must be taken for the transmittal package to work properly. For example, if SHX fonts are detected in one of the transmittal drawings, you are instructed where to copy these files so that they can be detected on the system where the transmittal package is being installed. If you have created a text file of default notes, the notes are also included in the report.

**Save As** Opens a File Save dialog box, in which you can specify a location in which to save a report file. Note that a report file is automatically included with all transmittal packages that you generate; by choosing Save As, you can save an additional copy of a report file for archival purposes.

## Transmittal Setups Dialog Box

### Quick Reference

File ➤ eTransmit
**etransmit**




Creates, modifies, and deletes transmittal setups.

### Transmittal Setups

Lists transmittal setups.

#### New

Creates a new transmittal setup.

**New Transmittal Setup Name** Enter the name of the new transmittal setup.

**Based On** Select an existing transmittal setup from which the new one will be created.

#### Rename

Renames the currently highlighted transmittal setup.

#### Modify

Displays the Modify Transmittal Setup dialog box (page 547), in which you can specify options for the transmittal package.

#### Delete

Removes the currently highlighted transmittal setup.

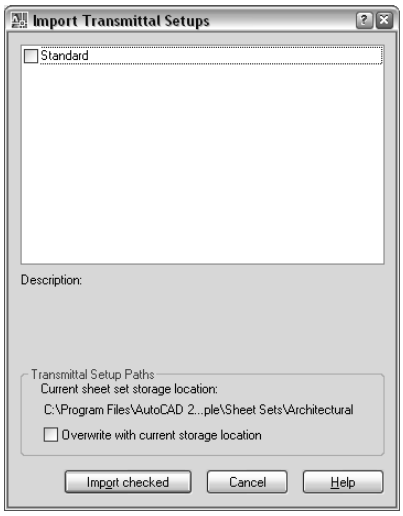
#### Import

In a sheet set context, opens standard file selection dialog box (page 931), in which you can navigate to a sheet set data (DST) file. Then displays the Import Transmittal Setups dialog box (page 545), in which you can specify the transmittal setups that you want to import.

## Import Transmittal Setups Dialog Box

### Quick Reference

File ➤ eTransmit
**etransmit**




Imports selected transmittal setups from a specified sheet set data file.

### List of Transmittal Setups

Displays a list of transmittal setups. Check the ones that you want to import into the current sheet set.

### Description

Displays the optional description that is stored with the selected transmittal setup.

### Current Sheet Set Storage Location

Displays the storage location for the current sheet set.

### Overwrite with Current Storage Location

When checked, replaces the imported sheet set storage locations with the storage location of the current sheet set.

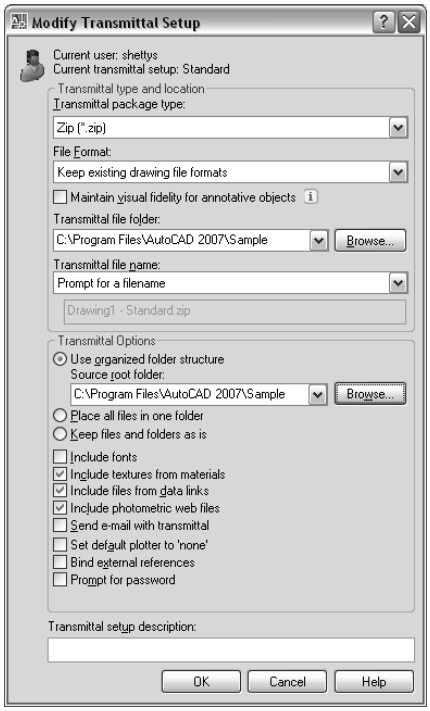
### Import Checked

Imports the transmittal setups on the list that display check marks.

# Modify Transmittal Setup Dialog Box

## Quick Reference

File ➤ eTransmit
**etransmit**




### Transmittal Package Type

Specifies the type of transmittal package created.

**Folder** Creates a transmittal package of uncompressed files in a new or existing folder.

**Self-Extracting Executable** Creates a transmittal package of files as a compressed, self-extracting executable file. Double-clicking the resulting EXE file decompresses the transmittal package and restores the files to the folder location that you specify.

**Zip** Creates a transmittal package of files as a compressed ZIP file. To restore the files to the folder location that you specify, you need a decompression utility such as the shareware application PKZIP or WinZip.

### File Format

Specifies the file format to which all drawings included in a transmittal package will be converted. You can select a drawing file format from the drop-down list.

### Maintain Visual Fidelity for Annotative Objects

Specifies whether or not drawings are saved with visual fidelity for objects. Click the information icon to learn more about saving drawings with visual fidelity.

Annotative objects may have multiple . Annotative objects are decomposed and scale representations are saved to separate layers, which are named based on their original layer and appended with a number.

### Transmittal File Folder

Specifies the location in which the transmittal package is created. Lists the last nine locations in which transmittal packages were created. To specify a new location, click Browse and navigate to the location you want.

If this field is left unchanged, the transmittal file is created in the folder containing the first specified drawing file. In a sheet set context, the transmittal file is created in the folder containing the sheet set data (DST) file.

### Browse

Opens a standard file selection dialog box, in which you can navigate to a location where you create the transmittal package.

### Transmittal File Name

Specifies the method for naming the transmittal package. Displays the default file name for the transmittal package. This option is not available if the transmittal package type is set to Folder.

**Prompt for a File Name** Displays a standard file selection dialog box where you can enter the name of the transmittal package.

**Increment File Name if Necessary** Uses a logical default file name. If the file name already exists, a number is added to the end. This number is incremented each time a new transmittal package is saved.

**Overwrite if Necessary** Uses a logical default file name. If the file name already exists, the existing file is automatically overwritten.

### Transmittal Options

Provides options for organizing the files and folders that are included in the transmittal package.

**Use Organized Folder Structure** Duplicates the folder structure for the files being transmitted. The root folder is the top-level folder within a hierarchical folder tree.

The following considerations apply:

- Relative paths remain unchanged. Relative paths outside the source root folder retain up to one level of the folder path above them, and are placed in the root folder.
- Full (absolute) paths within the root folder tree are converted to relative paths. Full paths outside the source root folder retain up to one level of the folder path above them, and are placed in the root folder.
- Full paths outside the root folder tree are converted to No Path and are moved to the root folder or to a folder within the root folder tree.
- A *Fonts* folder is created, if necessary.
- A *PlotCfgs* folder is created, if necessary.
- A *SheetSets* folder is created to hold all support files for sheet sets, if necessary. The sheet set data (DST) file, however, is placed in the root folder.

This option is not available if you're saving a transmittal package to an Internet location.

**Source Root Folder** Defines the source root folder for relative paths of drawing-dependent files, such as xrefs.

The source root folder also contains the sheet set data (DST) file when a sheet set is transmitted.

**Browse** Opens a standard file selection dialog box, in which you can navigate to specify a source root folder.

**Place All Files in One Folder** All files are installed to a single, specified target folder when the transmittal package is installed.

**Keep Files and Folders As Is** Preserves the folder structure of all files in the transmittal package, facilitating ease of installation on another system. This option is not available if you're saving a transmittal package to an Internet location.

**Include Fonts** Includes any associated font files (TTF and SHX) with the transmittal package.

---

**NOTE** Because TrueType fonts are proprietary, you should make sure that the recipient of the transmittal package also owns the TrueType fonts. If you are not sure whether the recipient owns the TrueType fonts, clear this option. If any required TrueType fonts are not available to the recipient, the font specified by the FONTALT system variable is substituted.

---

**Include Textures from Materials** Includes textures from materials that are attached to objects or faces.

**Include Files from Data Links** Adds external files referenced by a data link to the transmittal package.

**Include Photometric Web Files** Includes photometric web files that are associated with web lights in the drawing.

**Send E-mail with Transmittal** Launches the default system email application when the transmittal package is created so that you can send an email that includes the transmittal package as an attachment.

**Set Default Plotter to 'None'** Changes the printer/plotter setting in the transmittal package to None. Your local printer/plotter settings are usually not relevant to the recipient.

**Bind External References** Binds all external references to the files to which they were attached.

**Prompt for Password** Opens the Transmittal - SetPassword dialog box (page 551), where you can specify a password for your transmittal package.

**Include Drawing Set Data and Files** Includes the sheet set data (DST) file, callout and label block (DWG) files, and the associated drawing template (DWT) file with the transmittal package.

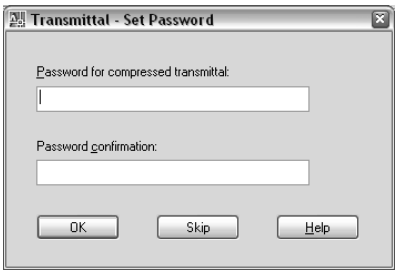
### Transmittal Setup Description

Enter a description for the transmittal setup. This description is displayed in the Create Transmittal dialog box below the list of transmittal file setups. You can select any transmittal setup in the list to display its description.

# Transmittal - Set Password Dialog Box

## Quick Reference

File ➤ eTransmit
**etransmit**




### Password for Compressed Transmittal

Provides a space for an optional password for the transmittal package. When others attempt to open the transmittal package, they must provide this password to access the files. Password protection cannot be applied to folder transmittal packages.

### Password Confirmation

Provides a space to confirm the password that you entered in the Password field. If the two passwords do not match, you are prompted to reenter them.

# -ETRANSMIT

## Quick Reference

If you enter **-etransmit** at the command prompt, the following ETRANSMIT command prompts are displayed.

Enter an option [Create transmittal package/Report only/CUrrent setup/CHoose setup/Sheet set> <Report only>: *Enter an option*
**Report Only** Creates a report (TXT) file without creating a transmittal package. This report file is based on the current transmittal setup.

**Current Setup** Displays the name of the current transmittal setup.

**Choose Setup** Specifies the transmittal setup to use for the transmittal package.

Choose Transmittal Setup or [?] <*current*>: *Enter a predefined transmittal setup name, enter* **?** *to display a list of transmittal setups, or press* ENTER

**Sheet Set** Specifies a sheet set and transmittal setup to use for the transmittal package. This option is available only when a sheet set is open.

Sheet Set name or [?] <*current*>: *Enter a predefined sheet set name, enter* **?** *to display a list of sheet sets, or press* ENTER

Choose Transmittal Setup or [?] <*current*>: *Enter a predefined transmittal setup name, enter* **?** *to display a list of transmittal setups, or press* ENTER

# EXPLODE

## Quick Reference

Breaks a compound object into its component objects

Modify 
Modify ➤ ExplodeAt the Command prompt, enter explode.
**explode**
Select objects: *Use an object selection method and press* ENTER *when you finish*

The color, linetype, and lineweight of any exploded object might change. Other results differ depending on the type of compound object you're

exploding. See the following list of objects that can be exploded and the results for each.

To explode objects and change their properties at the same time, use *XPLODE*.

---

**NOTE** If you're using a script or an ObjectARX® function, you can explode only one object at a time.

---

**2D and Lightweight Polyline** Discards any associated width or tangent information. For wide polylines, the resulting lines and arcs are placed along the center of the polyline.

**3D Polyline** Explodes into line segments. Any linetype assigned to the 3D polyline is applied to each resulting line segment.

**3D Solid** Explodes planar faces into regions. Nonplanar faces explode into surfaces.

**Annotative Objects** Explodes the current scale representation into its constituent parts which become non. Other scale representations are removed.

**Arc** If within a nonuniformly scaled block, explodes into elliptical arcs.

**Block** Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal *X*, *Y*, and *Z* scales explode into their component objects. Blocks with unequal *X*, *Y*, and *Z* scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a "*E" prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

Blocks inserted with *MINSERT* and external references (xrefs) and their dependent blocks cannot be exploded.

**Body** Explodes into a single-surface body (nonplanar surfaces), regions, or curves.

**Circle** If within a nonuniformly scaled block, explodes into ellipses.

**Leaders** Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), multiline text, or tolerance objects, depending on the leader.

**Multiline Text** Explodes into text objects.

**Multiline** Explodes into lines and arcs.

**Polyface Mesh** Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

**Region** Explodes into lines, arcs, or splines.

# EXPORT

## Quick Reference

Saves objects to other file formats

File ➤ ExportAt the Command prompt, enter export.
**export**

The Export Data dialog box (a standard file selection dialog box (page 931)) is displayed.

In File Type, select the desired export format. In File Name, enter the name of the file to create. The objects are exported to the specified file format using the specified file name.

---

**NOTE** The Export Data dialog box records the last used file format selection and stores it for use during the current drawing session and between drawing sessions.

---

The following output types are available:

- *3D DWF (*.dwf):* Autodesk Design Web Format (see *3DDWF*)
- *Metafile (*.wmf):* Microsoft Windows® Metafile (see *WMFOUT*)
- *ACIS (*.sat):* ACIS solid object file (see *ACISOUT*)
- *Lithography (*.stl):* Solid object stereolithography file (see *STLOUT*)
- *Encapsulated PS (*.eps):* Encapsulated PostScript file
- *DXX Extract (*.dxx):* Attribute extract DXF™ file (see *ATTEXT*)
- *Bitmap (*.bmp):* Device-independent bitmap file (see *BMPOUT*)

- *Block (*.dwg):* drawing file (see *WBLOCK*)
- *V8 DGN (*.dgn):* MicroStation DGN file (see *DGNEXPORT*)

# EXPORTTOAUTOCAD

## Quick Reference

Creates a new DWG file with all AEC objects exploded

**-exporttoautocad** or **aectoacad**
Export options [Format (page 555)/Bind (page 555)/bind Type (page 556)/Maintain (page 556)/Prefix (page 556)/Suffix (page 556)/? (page 557)] <Enter for filename (page 555)>: *Enter an option or press* ENTER *to specify a file name*

You can create a new version of a drawing file with all proxy AEC objects exploded into basic AutoCAD objects. The new version of the drawing loses the intelligence of the custom AEC objects, but the resulting basic objects can be displayed and accessed in earlier versions of AutoCAD when object enablers are not available for those versions.

**NOTE** Any subsequent changes you make to this new drawing file do not affect the original drawing file.

### Enter for Filename

Creates a new drawing file with the specified name.

**NOTE** You can use the Prefix option to create a unique file name for this drawing and to prevent overwriting the existing drawing file.

Export drawing name <*current*>: *Enter a file name*

### Format

Determines the drawing file format for the resulting drawing file.

Enter file format [r14/2000/2004/2007] <2007>: *Specify the drawing file format*

### Bind

Determines how xrefs are treated when creating the new drawing.

Bind xrefs [Yes/No] <Yes>: *Enter* **y** *or* **n**
**Yes** Binds all xrefs to the drawing. Layers and other xref-dependent named objects are merged into the new drawing.

**No** Maintains the xrefs as links to other drawings.

### Bind Type

Determines how xref-dependent objects are treated when the Bind option is turned on.

Bind type [Bind/Insert] <Insert>: *Enter an option*
**Bind** Maintains the names of layers and other xref-dependent objects when binding xrefs.

**Insert** Merges the names of xref-dependent objects into the new drawing without including the original file name.

### Maintain

Determines how blocks within custom AEC objects are treated.

Maintain resolved properties [Yes/No] <Yes>: *Enter* **y** *or* **n**
**Yes** Explodes all block instances within custom AEC objects into basic AutoCAD objects. The resulting basic objects revert to their original properties rather than the properties of the block definitions.

**No** Does not explode any block instances within custom AEC objects. The properties of the block instances are determined in the usual way, depending on how the objects in the blocks were created and the property settings of the layers on which the blocks are inserted.

### Prefix

Specifies the prefix to be added to the current drawing file name.

Filename prefix <>: *Enter the characters to be added to the beginning of the current file name*

### Suffix

Specifies the suffix to be added to the current drawing file name.

Filename suffix <>: *Enter the characters to be added to the end of the file name*

#### ? List Settings

Lists the current settings for the command.

File format: *current setting*
Bind xrefs: *current setting*
Bind type: *current setting*
Filename prefix: *current setting*
Filename suffix: *current setting*

# EXTEND

## Quick Reference

Extends an object to meet another object

Modify 
Modify ➤ ExtendAt the Command prompt, enter extend.
**extend**
Current settings: Projection = *current*, Edge = *current*
Select boundary edges...
Select objects (page 557) or <select all>: *Select one or more objects and press* ENTER*, or press* ENTER *to select all displayed objects*
Select object to extend (page 557) or shift-select to trim (page 558) or [Fence (page 558)/Crossing (page 558)/Project (page 558)/Edge (page 560)/Undo (page 560)]: *Select objects to extend, or hold down* SHIFT *and select an object to trim, or enter an option*

#### Boundary Object Selection

Uses the selected objects to define the boundary edges to which you want to extend an object.

#### Object to Extend

Specifies the objects to extend. Press ENTER to end the command.

### Shift-Select to Trim

Trims the selected objects to the nearest boundary rather than extending them. This is an easy method to switch between trimming and extending.

### Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Specify first fence point: *Specify the starting point of the selection fence*
Specify next fence point or [Undo]: *Specify the next point of the selection fence or enter* **u**
Specify next fence point or [Undo]: *Specify the next point of the selection fence enter* **u**, *or press* ENTER

### Crossing

Selects objects within and crossing a rectangular area defined by two points.

Specify first corner: *Specify a point*
Specify opposite corner: *Specify a point at a diagonal from the first point*

---

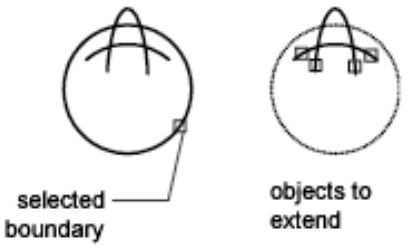
**NOTE** Some crossing selections of objects to be extended are ambiguous. EXTEND resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

---

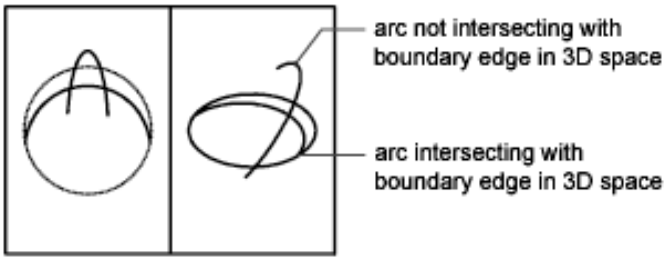
### Project

Specifies the projection method used when extending objects.

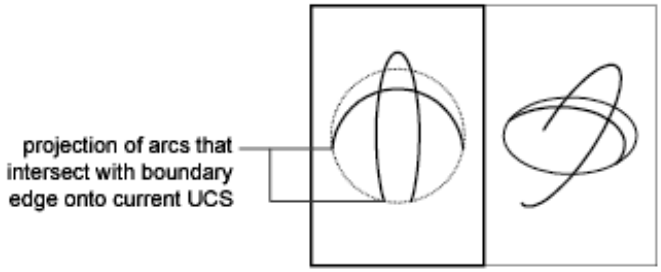
Enter a projection option [None/Ucs/View] <*current*>: *Enter an option or press* ENTER



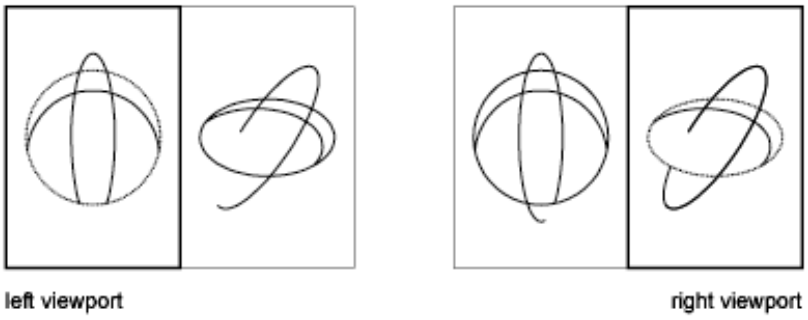
**None** Specifies no projection. Only objects that intersect with the boundary edge in 3D space are extended.




**UCS** Specifies projection onto the *XY* plane of the current user coordinate system (UCS). Objects that do not intersect with the boundary objects in 3D space are extended.

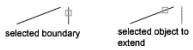



**View** Specifies projection along the current view direction.



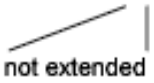
### Edge

Extends the object to another object's implied edge, or only to an object that actually intersects it in 3D space.




Enter an implied edge extension mode [Extend/No extend] <*current*>: *Enter an option or press* ENTER
**Extend** Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.


**No Extend** Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.




### Undo

Reverses the most recent changes made by EXTEND.

# EXTERNALREFERENCES

## Quick Reference

Displays the External References palette

Reference 
Insert ➤ External ReferencesAt the Command prompt, enter externalreferences.
**externalreferences**

The EXTERNALREFERENCES command opens the External References palette (page 561) if the palette is inactive. If the External References palette is active

but hidden, executing EXTERNALREFERENCES will open the palette. If you click anywhere outside of the External References palette, the palette returns to its auto-hidden state.

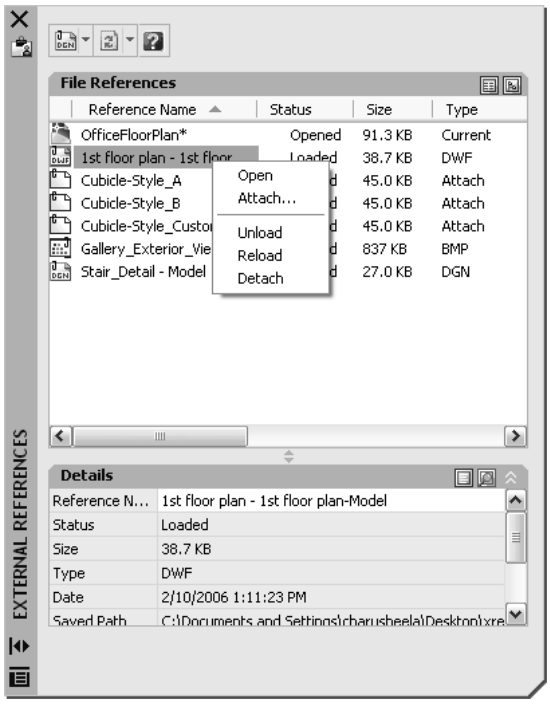
**NOTE** The *FILEDIA* system variable is ignored when attaching files from the External References palette.

# External References Palette

## Quick Reference

Reference
Insert ➤ External ReferencesAt the Command prompt, enter externalreferences.
**externalreferences**

The External References palette organizes, displays, and manages *referenced files*, such as referenced drawings (Xrefs), attached DWF or DGN underlays, and imported raster images.




**NOTE** When using the External References palette, it is recommended that you turn on the Auto-hide feature or anchor the palette. The palette will then hide automatically when you specify the insertion point of the external reference.

The External References palette contains a set of tool buttons (page 563), two dual mode data panes and a messaging field. The upper data pane, called the File References Pane (page 564), can be set to display file references in a list mode or tree mode. Shortcut menus and function keys provide options for working with the files. (page 566)The lower data pane, the Details / Preview Pane (page 569), can display properties for the selected file references or it can display a thumbnail preview of the selected file reference. The messaging field at the bottom presents information about selected file references that is pertinent under certain conditions.

# External References Palette Tool Buttons

## Quick Reference

The tool buttons at the top of the External References palette give you control over the types of files you can attach to the drawing and the ability to refresh the status of file references you already have attached.

Attach File 

The first button at the top of the External References palette allows you to attach DWG, DWF or raster images. The initial default state of the button is Attach DWG. The button retains the last type of attachment action used. Therefore, if you attach a DWF file, the button state will remain set to Attach DWF until you attach another file type.

### Attach File Drop Down Menu

The Attach File drop down menu displays a list of file types that you can attach. The menu contains:

- Attach DWG - Starts the *XATTACH* command.
- Attach Image - Starts the *IMAGEATTACH* command.
- Attach DWF - Starts the *DWFATTACH* command.
- Attach from Vault - Provides access to content stored in the Vault client.

---

**NOTE** Attach From Vault is only shown when the Vault client is installed. You must be an Autodesk Subscription customer to have access to the Vault client.

---

Refresh 

Resynchronizes the status data of referenced drawing files with the data in memory. Refresh interacts primarily with Autodesk Vault.

### Refresh Drop Down Menu

The Refresh drop down menu displays two choices:

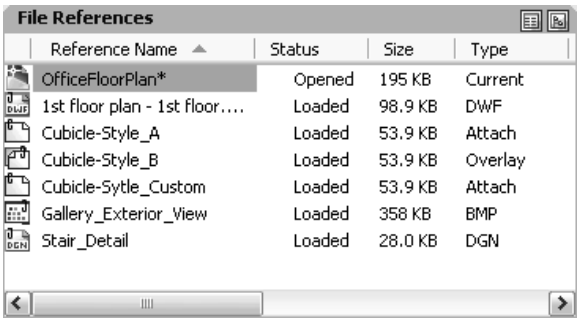
- Refresh

■ Reload All References

# File References List View / Tree View Pane

## Quick Reference

The upper File References pane can be set to display a listing of all the external references you have attached to your drawing. You can set the display to show your attachments in a list or as a tree structure. The default display mode of the File References pane is *list view*.




### List View

When the File References pane is set to list view, you are presented with a list of all the external references you have associated with the drawing. In list view, you can select multiple file references. The listed information includes the reference name, status, file size, file type, creation date and saved path.

**Reference Icons** Each file reference is preceded by an icon specific to that reference type.

 Indicates the current drawing icon. It represents the master drawing to which all external references are attached.

 Indicates a DWF Underlay attachment.

 Indicates a DWG (xref) attachment.

 Indicates a DWG (xref) overlay.

 Indicates a DGN underlay.

 Indicates a raster image attachment.

**Reference Name** The reference name column always displays the current drawing as the first entry, followed by additional attached files that are listed in the order they were attached.

**Status** The status of the referenced file:

- *Loaded* - The referenced file is currently attached to the drawing.
- *Unloaded* - The referenced file is marked to be unloaded from the drawing.
- *Not Found* - The referenced file is no longer exists in the valid search paths.
- *Unresolved* - The referenced file cannot be read.
- *Orphaned* - The referenced file is attached to another file that has an *Unresolved* status.

**Size** The size of the attached file reference.

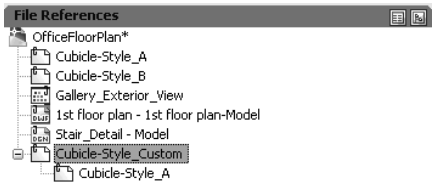
**Type** The file type of the referenced file. Drawing (Xref) files display as attachments or overlays, raster images show their file format, and DWF underlays are listed as their respective file type.

**Date** The date when the referenced file was created or last saved.

**Saved Path** Displays the path that is saved with the drawing when the referenced file is attached.

### Tree View 

The tree view mode of the File References pane displays all the referenced file definitions and the levels of nesting of file references within xrefs.




The top level of the tree view always shows the current drawing. Referenced files appear at the next level. Referenced files that contain their own nested file references can be opened to show deeper levels. When making selections in tree view, only one file reference can be selected at a time.

# File References Pane Shortcut Menus and Function Keys

## Quick Reference

When working in the File References pane, there are several shortcut menus that can be displayed when you right-click on file references or empty areas of the pane. The following tables show the shortcut menu items that you are presented under certain conditions.

### Nothing Selected

When no file references are selected, right-clicking to open the shortcut menu presents the following functions:

| Menu Item | Description |
|---|---|
| Reload All References | Reloads all referenced files. (Unavailable if no file references are attached) |
| Select All | Selects all file references, excluding the current drawing. This item is not displayed in tree view. |
| Attach DWG | Starts the *XATTACH* command. |

| Menu Item | Description |
|---|---|
| Attach Image | Starts the *IMAGEATTACH* command. |
| Attach DWF | Starts the *DWFATTACH* command. |
| Attach DGN | Starts the *DGNATTACH* command. |
| Attach from Vault | Launches the Vault Attach File dialog box - Only displayed when the Vault Client is installed. |
| Log In | Allows you to log into the Vault Server. If you are already logged in, this item is unavailable - Only displayed when the Vault Client is installed. |
| Log Out | Allows you to log out of the Vault Server. If you are already logged out, this item is unavailable - Only displayed when the Vault Client is installed. |
| Close | Closes the External References palette. |

When you select a file reference, right-clicking to open the shortcut menu presents the following functions:

| Menu Item | Description | Reference Status |
|---|---|---|
| Open | Opens the selected file reference in the application specified by the operating system. | Available only for file references with a Loaded status - Unavailable when Unloaded, Not Found or Unresolved. |
| Attach | Opens the dialog box corresponding to the selected reference type.<br>■ Selecting a DWG reference opens the External Reference dialog box (page 1568). | Always available - status has no affect on this function. |

| Menu Item | Description | Reference Status |
|---|---|---|
| | ■ Selecting a raster image reference opens the Image dialog box (page 514).<br>■ Selecting a DWF reference opens the Attach DWF Underlay dialog box (page 514). | |
| Unload | Unloads the selected file references. | Always available - status has no affect on this function. |
| Reload | Reloads the selected file references. | Always available - status has no affect on this function. |
| Detach | Detaches the selected file references. | Always available - status has no affect on this function. |
| Bind | Displays the Bind Xrefs dialog box (page 1582). The selected DWG reference is bound to the current drawing - Only available for referenced DWG files. | Available only for file references with a Loaded status - Unavailable when Unloaded, Not Found or Unresolved. |
| Check In | Returns a modified file that is checked out of the Vault. The previous version is retained in the file history - Only displayed when the Vault Client is installed. | Determined by Vault functionality. |
| Check Out | Retrieves a read/write copy of a file stored in the Vault - Only displayed when the Vault Client is installed. | Determined by Vault functionality. |
| Undo Check Out | Releases a file that you have checked out of the Vault - Only displayed when the Vault Client is installed. | Determined by Vault functionality. |

### Function Key Access

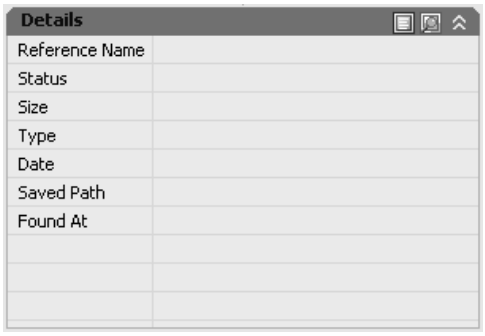
Some tasks in the File References pane can be accessed with function keys.

- F2 - Accesses the Rename function when a single file reference is selected.
- F3 - List View toggle.
- F4 - Tree View toggle.

# Details / Preview Pane

## Quick Reference

The lower data pane of the External References palette can be set to display file reference properties or a preview image of the selected file reference.




### Details Pane 

When displaying the Details mode, properties for the selected file reference are reported. Each file reference has a core set of properties and some file references, like referenced images, display properties specific to the file type. The core set of details include the reference name, status, file size, file type, creation date, saved path, found at path and file version, if the Vault client is installed. Some of the properties can be edited.

**Reference Name** Displays the file reference name. This property can be edited only if single file references are selected. The reference name shows **Varies** if multiple file references are selected. This property is editable for all the file references.

**Status** Shows whether the file reference is loaded, unloaded or not found. This property cannot be edited.

**Size** Shows the file size of the selected file reference. The size is not displayed for file references that are unloaded or not found. This property cannot be edited.

**Type** Indicates whether the file reference is an attachment or overlay, the type of image file or DWF underlay. This property cannot be edited.

**Date** Displays the last date the file reference was modified. This date is not displayed if the file reference is unloaded or not found. This property cannot be edited.

**Saved Path** Shows the saved path of the selected file reference (this is not necessarily where the file reference is found). This property cannot be edited.

**Found At** Displays the full path of the currently selected file reference. This is where the referenced file is actually found and is not necessarily the same as the saved path. Clicking the […] button displays the Select Image File dialog box where you can select a different path or file name. You can also type directly into the path field. These changes are stored to the Saved Path property if the new path is valid.

**File Version** File Version property defined by the Vault client. This property is only displayed when you are logged into the Vault.

### Specific Image Properties

If you select a referenced image, additional properties are displayed. None of the added image properties can be edited.

**Color System** Displays the color system.

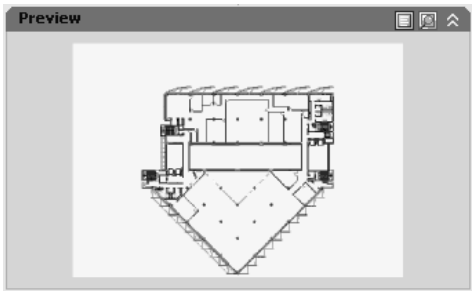
**Color Depth** The amount of information that is stored in each pixel of a raster image. Higher color depth values produce smoother degrees of shading.

**Pixel Width** The width of the raster image measured in pixels.

**Pixel Height** The height of the raster image measured in pixels.

**Resolution** The width and height resolution in dots per inch (dpi).

**Default Size (in AutoCAD units)** The width and height of the raster image measured in AutoCAD units.




**Preview Pane** 

The Preview mode displays a preview for the file reference selected in the File References pane.

The preview image is displayed only when a single file reference is selected from the File References pane. There are no other controls for this data pane. When no reference file is selected, the preview pane displays a solid grey field. If there is no preview available, the text "Preview not available" is displayed in the center of the pane.

### Messaging Field

Below the Details / Preview pane is a messaging field that supplies information about certain selected file references. When you select one or more nested references, information is displayed regarding the file references. Messages also appear if you decide the change the name of a file reference.

# EXTERNALREFERENCESCLOSE

## Quick Reference

Closes the External References palette

**externalreferencesclose**

The EXTERNALREFERENCESCLOSE command closes the External References palette (page 561). If the External References palette is currently displayed, either in an auto-hidden state or open state, it is closed.

# EXTRUDE

## Quick Reference

Creates a 3D solid or surface by extruding an object or planar face a specified distance and direction

Modeling 
Draw ➤ Modeling ➤ ExtrudeAt the Command prompt, enter extrude.
**extrude**
3D Make panel, Extrude
Current wire frame density: ISOLINES=4
Select objects to extrude: (page 572)
Specify height of extrusion (page 573) or [Direction (page 574)/Path (page 574)/Taper angle (page 575)] <default>: *Specify a distance or enter* **p**

You can select the objects to extrude before you start the command.

The *DELOBJ* system variable controls whether the object(s) and path (if selected) are automatically deleted when the solid or surface is created or whether you are prompted to delete the object(s) and path.

### Object Selection


Specifies the objects to extrude. You can extrude the following objects and subobjects:

- Lines
- Arcs
- Elliptical arcs
- 2D polylines
- 2D splines
- Circles

- Ellipses
- 2D solids
- Traces
- Regions
- Planar 3D polylines
- Planar 3D faces
- Planar surfaces
- Planar faces on solids

**NOTE** You can select faces on solids by pressing and holding CTRL, and then selecting these subobjects.

You cannot extrude objects contained within a block or polylines that have crossing or self-intersecting segments.

If a selected polyline has width, the width is ignored and the polyline is extruded from the center of the polyline path. If a selected object has thickness, the thickness is ignored.

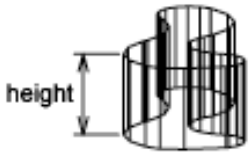
**NOTE** You can use *CONVTOSOLID* to convert polylines and circles with thickness to solids. You can use *CONVTOSURFACE* to convert lines with thickness; arcs with thickness; and open, zero-width polylines with thickness to surfaces.

### Height of Extrusion

Extrudes objects along the positive *Z* axis of the object's coordinate system if you enter a positive value. If you enter a negative value, the objects are extruded along the negative *Z* axis. Objects do not have to be parallel to the same plane. If all the objects are on a common plane, the objects are extruded in the direction of the normal of the plane.

By default, planar objects are extruded in the direction of the object's normal.

Specify second point: *Specify a point*



### Direction

Specifies the length and direction of the extrusion with two points you specify.

Specify start point of direction: *Specify a point*
Specify end point of direction: *Specify a point*

### Path

Selects the extrusion path based on a specified object. The path is moved to the centroid of the profile. Then the profile of the selected object is extruded along the chosen path to create solids or surfaces.


Select extrusion path: *Use an object selection method*

The following objects can be paths:

- Lines
- Circles
- Arcs
- Ellipses
- Elliptical arcs
- 2D polylines
- 3D polylines
- 2D splines
- 3D splines
- Edges of solids
- Edges of surfaces
- Helixes

**NOTE** You can select faces and edges on solids by pressing and holding CTRL, and then selecting these subobjects.

The path should not lie on the same plane as the object, nor should the path have areas of high curvature.

The extruded solid starts from the plane of the object and maintains its orientation relative to the path.

If the path contains segments that are not tangent, the program extrudes the object along each segment and then miters the joint along the plane bisecting the angle formed by the segments. If the path is closed, the object should lie on the miter plane. This allows the start and end sections of the solid to match up. If the object is not on the miter plane, the object is rotated until it is on the miter plane.

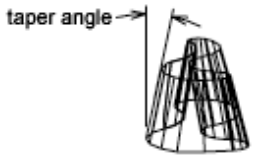
Objects with multiple loops are extruded so that all the loops appear on the same plane at the end section of the extruded solid.

### Taper Angle

Specify angle of taper for extrusion <0>: *Specify an angle between -90 and +90 degrees, press* ENTER, *or specify a point*

If you specify a point for the taper angle rather than enter a value, you must pick a second point. The taper angle applied to the extrusion is the distance between the two specified points.

Specify second point: *Specify a point*




Positive angles taper in from the base object. Negative angles taper out. The default angle, 0, extrudes a 2D object perpendicular to its 2D plane. All selected objects and loops are tapered to the same value.

Specifying a large taper angle or a long extrusion height can cause the object or portions of the object to taper to a point before reaching the extrusion height.

Individual loops of a region are always extruded to the same height.

When an arc is part of a tapered extrusion, the angle of the arc remains constant, and the radius of the arc changes.

# F Commands

# 7

**In this chapter**

- FIELD
- FILL
- FILLET
- FILTER
- FIND
- FLATSHOT
- FOG
- FREESPOT
- FREEWEB

# FIELD

## Quick Reference

Creates a multiline text object with a field that can be updated automatically as the field value changes

Insert ➤ FieldAt the Command prompt, enter Field.
Right-click while any text command is active, and click Insert Field.
**field**

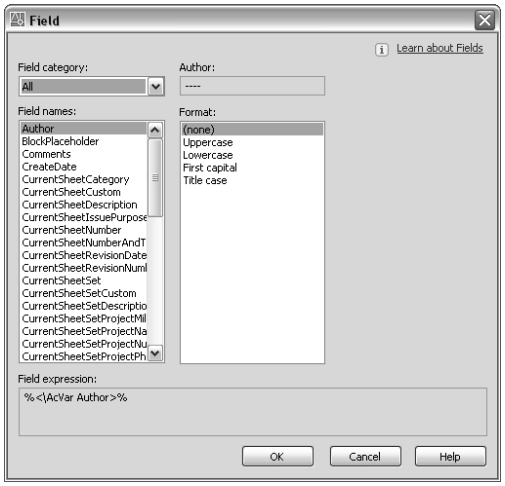
The Field dialog box (page 578) is displayed.

# Field Dialog Box

## Quick Reference

Insert ➤ FieldAt the Command prompt, enter Field.
Right-click while any text command is active, and click Insert Field.
**field**

Inserts a field in the drawing.




The options available in the Field dialog box change with the field category and field name.

**Field Category** Sets the types of fields to be listed under Field Names (for example, Date & Time, Document, and Objects). Other includes DieselExpression, LispVariable, and SystemVariable.

**Field Names** Lists the fields that are available in a category. Select a field name to display the options available for that field.

*Field Value* Displays the current value of the field, or displays an empty string (----) if the field value is invalid.

The label for this item changes with the field name. For example, when Filename is selected in the Field Names list, the label is Filename and the value is the name of the current drawing file. The label is Property for object fields. Exception: when a date field is selected, the selected date format is displayed; for example, M/d/yyyy.

*Format List* Lists options for display of the value of the field. For example, date fields can display the name of the day or not, and text strings can be uppercase, lowercase, first capital, or title case. The value displayed in the Fields dialog box reflects the format that you select.

**Field Expression** Displays the expression that underlies the field. The field expression cannot be edited, but you can learn how fields are constructed by reading this area.

### Options for Sheet Set and Sheet View Fields

The following additional options are available when sheet-set-related fields are selected:

**Sheet Set** Specifies the name of the sheet set.

**Sheet Navigation Tree** Displays a tree view of sheets or sheet views from the Sheet Set Manager, from which you can select an item for the field.

**Property** Displays the properties available as fields for the item selected in the tree.

**Associate Hyperlink** Adds a hyperlink to the field when it is created. You can use CTRL+click to jump to the sheet or view. This option is not available for the ViewportScale field.

### Options for BlockPlaceholder Fields

BlockPlaceholder fields are only available for insertion in the Attribute Definition dialog box when the Block Editor (page 160) is open.

The following additional options are available when the BlockPlaceholder field is selected:

**Block Name** Displays the name of the current block definition.

**Temporary Value** Displays the value for the placeholder field in the selected format.

**Block Reference Properties** Displays a list of block reference properties for the current block definition.

### Options for SheetSetPlaceholder Fields

The following additional options are available when the SheetSetPlaceholder field is selected:

**Placeholder Type** Displays a list of available placeholder fields.

**Temporary Value** Displays the value for the placeholder field in the selected format.

For example, with SheetSet Placeholder selected in the Field Names list, SheetTitle selected in the Placeholder Type list, and Uppercase selected in the Format list, Temporary Value displays SHEETTITLE. When the drawing is placed in a sheet set, this field displays the title of the sheet.

### Options for Fields in the Objects Field Category

The following additional options are available when object fields are selected:

**Named Object Type/Object Type** When NamedObject is selected in Field Names, lists the types of named objects in the drawing. When Object is selected, displays the type of object selected. Use the Select Object button to temporarily close the dialog box and select an object in the drawing.

**Property/Name** When NamedObject is selected in Field Names, lists the names of all the objects in the drawing of the selected type. When Object is selected in Field Names, lists the properties of the selected object that are available as fields. When a block with attributes is selected, the attribute names are displayed in the list of properties.

**Formula** When Formula is selected in Field Names, provides a place for creating a formula to insert in text or in a table cell.

**Average/Sum/Count** When Formula is selected in Field Names, closes the Field dialog box temporarily while you specify table cells. The result is appended to the formula.

**Cell** When Formula is selected in Field Names, closes the Field dialog box temporarily while you specify a table cell. The cell address is appended to the formula.

**Precision** Specifies precision for fields based on the selected format. Select Current Precision to use the current setting of the *LUPREC* system variable.

**Additional Format** Displays the Additional Format dialog box (page 1412).

**Evaluate** Updates the value in Preview when you have manually changed the text in Formula.

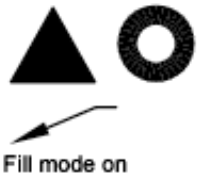
**Display Value for Block Reference** Specifies that the object property fields in a block reference evaluate the properties of nested objects relative to the block reference's size and orientation within the drawing, not within the block definition.

# FILL

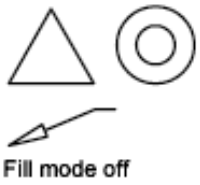
## Quick Reference

Controls the filling of objects such as hatches, two-dimensional solids, and wide polylines

**fill** (or '**fill** for transparent use)
Enter mode [ON (page 581)/OFF (page 581)] <*current*>: *Enter* **on** *or* **off**, *or press* ENTER
**On** Turns on Fill mode. For the filling of a 3D object to be visible, its extrusion direction must be parallel to the current viewing direction, and hidden lines must not be suppressed.




**Off** Turns off Fill mode. Only the outlines of objects are displayed and plotted. Changing Fill mode affects existing objects after the drawing is regenerated. The display of lineweights is not affected by the Fill mode setting.




# FILLET

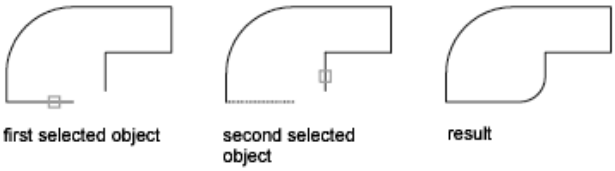
## Quick Reference

Rounds and fillets the edges of objects

Modify 
Modify ➤ FilletAt the Command prompt, enter fillet.
**fillet**
Current settings: Mode = *current*, Radius = *current*
Select first object (page 582) or [Undo (page 585)/Polyline (page 585)/Radius (page 585)/Trim (page 585)/Multiple (page 585)]: *Use an object selection method or enter an option*

### First Object

Selects the first of two objects required to define a 2D fillet or selects the edge of a 3D solid to round or fillet the edge.

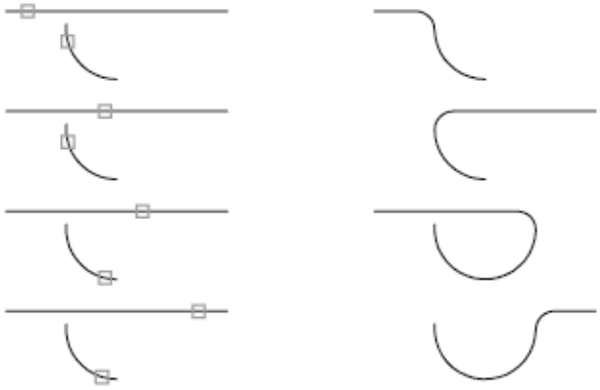



Select second object or shift-select to apply corner: *Use an object selection method or hold down SHIFT and select an object to create a sharp corner*

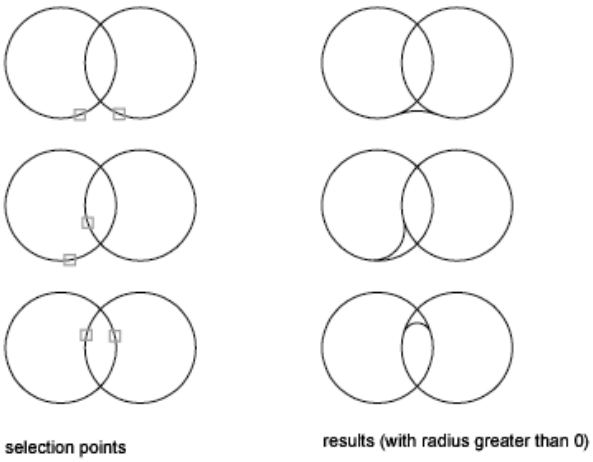
If you select lines, arcs, or polylines, their lengths adjust to accommodate the fillet arc. You can hold down SHIFT while selecting the objects to override the current fillet radius with a value of 0.

If the selected objects are straight line segments of a 2D polyline, they can be adjacent or separated by one other segment. If they are separated by another polyline segment, FILLET deletes the segment that separates them and replaces it with the fillet.

More than one fillet can exist between arcs and circles. Select the objects close to where you want the endpoints of the fillet.




FILLET does not trim circles; the fillet arc meets the circle smoothly.

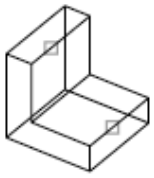


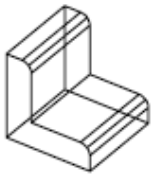

If you select a 3D solid, you can select multiple edges, but you must select the edges individually.

Enter fillet radius <*current*>: *Specify a distance or press* ENTER
Select an edge or [Chain/Radius]: *Select edge(s), enter* **c**, *or enter* **r**

### Edge

Selects a single edge. You can continue to select single edges until you press ENTER.


selecting edges

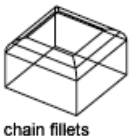

single edge fillets

If you select three or more edges that converge at a vertex to form the corner of a box, FILLET computes a vertex blend that is part of a sphere if the three incident fillets have the same radii.

### Chain

Changes from selection of single edges to selection of sequential tangent edges, called a *chain* selection.

Select edge chain or <Edge/Radius>: *Select an edge chain, enter* **e**, *or enter* **r**
**Edge Chain** Selects a tangential sequence of edges when you select a single edge. For example, if you select an edge on the top of a 3D solid box, FILLET also selects the other tangential edges on the top.


chain fillets

**Edge** Switches to a single-edge selection mode.

**Radius** Defines the radius of the rounded edge.

### Radius

Defines the radius of the rounded edge.

Enter fillet radius <*current*>: *Specify a distance or press* ENTER

The previous prompt is displayed:

Select an edge or [Chain/Radius]: *Select one or more edges, or enter* **c** *or* **r**

### Undo

Reverses the previous action in the command.

### Polyline

Inserts fillet arcs at each vertex of a 2D polyline where two line segments meet.

Select 2D polyline:

If one arc segment separates two line segments that converge as they approach the arc segment, FILLET removes the arc segment and replaces it with a fillet arc.


### Radius

Defines the radius of the fillet arc.

Specify fillet radius <*current*>: *Specify a distance or press* ENTER

The value you enter becomes the current radius for subsequent FILLET commands. Changing this value does not affect existing fillet arcs.

### Trim

Controls whether FILLET trims the selected edges to the fillet arc endpoints.

Enter Trim mode option [Trim/No trim] <*current*>: *Enter an option or press* ENTER
**Trim** Trims the selected edges to the fillet arc endpoints.

**No Trim** Does not trim the selected edges.

### Multiple

Rounds the edges of more than one set of objects. FILLET displays the main prompt and the Select Second Object prompt repeatedly until you press ENTER to end the command.

# FILTER

## Quick Reference

Creates a list of requirements that an object must meet to be included in a selection set

**filter** (or '**filter** for transparent use)

The Object Selection Filters dialog box (page 586) is displayed.

# Object Selection Filters Dialog Box

## Quick Reference

**filter** (or '**filter** for transparent use)

Selects, edits, and names filters for object selection.


### Filter Property List

Displays a list of the filter properties that compose the current filter. The current filter is the filter that you select in Current in the Named Filters area.

### Select Filter

Adds filter properties to the current filter.

**Object Types and Logical Operators** Lists object types that you can filter and logical operators (AND, OR, XOR, and NOT) for grouping the filter expressions.

If you use logical operators, make sure that you pair and balance them correctly in the filter list. The number of operands you can enclose depends on the operation.

**Logical operators**

| Starting operator | Encloses | Ending operator |
|---|---|---|
| Begin AND | One or more operands | End AND |
| Begin OR | One or more operands | End OR |
| Begin XOR | Two operands | End XOR |
| Begin NOT | One operand | End NOT |

For example, the following filter selects all circles except the ones with a radius greater than or equal to 1.0:

Object =Circle

**Begin NOT

Circle Radius >= 1.00

**End NOT

**X, Y, Z Parameters** Define additional filter parameters depending on the object. For example, if you select Line Start, you can enter the *X*, *Y*, and *Z* coordinate values that you want to filter.

In the filter parameters, you can use relative operators such as < (less than) or > (greater than). For example, the following filter selects all circles with center points greater than or equal to 1,1,0 and radii greater than or equal to 1:

Object = Circle

Circle Center X >= 1.0000 Y >= 1.0000 Z >= 0.0000

Circle Radius >= 1.0000

**Select** Displays a dialog box listing all items of the specified type in the drawing. Select the items to filter. For example, if you select the object type Color, Select displays a list of colors to choose for the filter.

**Add to List** Adds the current Select Filter property to the filter list. Filter properties that you add to the unnamed filter remain available during the current work session unless you manually delete them.

**Substitute** Replaces the filter property selected in the filter property list with the one displayed in Select Filter.

**Add Selected Object** Adds one selected object in the drawing to the filter list.

### Edit Item

Moves the selected filter property into the Select Filter area for editing. To edit a filter property, select it and choose Edit Item. Edit the filter property and choose Substitute. The edited filter replaces the selected filter property.

### Delete

Deletes a selected filter property from the current filter.

### Clear List

Deletes all the listed properties from the current filter.

### Named Filters

Displays, saves, and deletes filters.

**Current** Displays saved filters. Select a filter list to make it current. The named filter and its list of properties are loaded from the default file, *filter.nfl*.

**Save As** Saves a filter and its list of properties. The filter is saved in the *filter.nfl* file. Names can contain up to 18 characters.

**Delete Current Filter List** Deletes a filter and all its properties from the default filter file.

### Apply

Exits the dialog box and displays the Select Objects prompt, where you create a selection set. The current filter is used on the objects you select.

# FIND

## Quick Reference

Finds, replaces, selects, or zooms to specified text

Edit ➤ FindAt the Command prompt, enter find.
End any active commands, right-click in the drawing area, and choose Find.
**find**

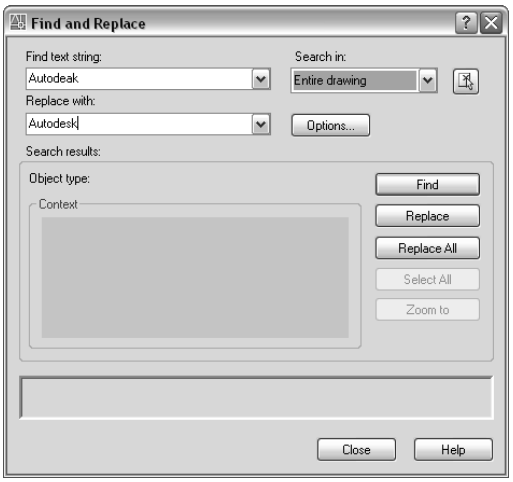
The Find and Replace dialog box (page 589) is displayed.

# Find and Replace Dialog Box

## Quick Reference

Edit ➤ Find
End any active commands, right-click in the drawing area, and choose Find.
**find**

Specifies the text you want to find, replace, or select and controls the scope and results of the search.



**Find Text String** Specifies the text string you want to find. Enter a text string including any wild-card characters, or choose one of the six most recently used strings from the list.

**Replace With** Specifies the text string you want to use to replace the found text. Enter a string, or choose one of the most recently used strings from the list.

**Search In** Specifies whether to search the entire drawing or only the current selection. If something is already selected, Current Selection is the default value. If nothing is selected, Entire Drawing is the default value. You can use the Select Objects button to temporarily close the dialog box and create or modify the selection set.

**Select Objects Button** Closes the dialog box temporarily so that you can select objects in your drawing. Press ENTER to return to the dialog box. When you select objects, Search In displays Current Selection.

**Options** Displays the Find and Replace Options dialog box (page 591), in which you can define the type of objects and words that you want to find.

**Find/Find Next** Finds the text that you enter in Find Text String. If you have not entered text in Find Text String, this option is not available. Found text is displayed in the Context area. Once you find the first instance of the text, the Find option becomes Find Next, which you can use to find the next instance.

**Replace** Replaces found text with the text that you enter in Replace With.

**Replace All** Finds all instances of the text that you enter in Find Text String and replaces it with the text in Replace With. The Search In setting controls whether to find and replace text in the entire drawing or text in the current selection. The status area confirms the replacements and indicates the number of replacements that were made.

**Select All** Finds and selects all loaded objects containing instances of the text that you enter in Find Text String. This option is available only when you set Search In to Current Selection. When you choose Select All, the dialog box closes and a message at the command prompt indicates the number of objects that it found and selected. Note that Select All does not replace text; any text in Replace With is ignored.

**Zoom To** Displays the area in the drawing that contains the results of a find or replace search. Although model space and all layouts defined for the drawing are searched, you can only zoom to text in the current Model or layout tab. When zooming to text found in a multiline text object, in some cases the found text string may not be displayed in a visible area of the drawing.

**Object Type** Specifies the type of object the text was found in.

**Context** Displays and highlights the currently found text string in its surrounding context. If you click Find Next, the Context area is refreshed and displays the next found text string in its surrounding context.

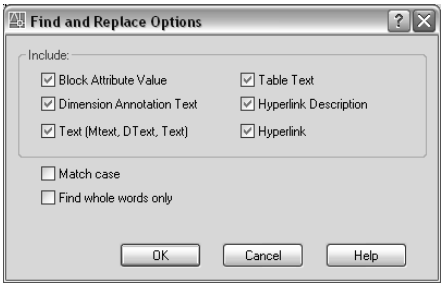
**Status** Displays confirmation of find and replace searches.

# Find and Replace Options Dialog Box

## Quick Reference

Edit ➤ FindAt the Command prompt, enter find.
End any active commands, right-click in the drawing area, and choose Find.
**find**

Defines the type of objects and words that you want to find.




**Include** Specifies the type of objects you want to include in the search. By default, all options are selected.

**Match Case** Includes the case of the text in Find Text String as part of the search criteria.

**Find Whole Words Only** Finds only whole words that match the text in Find Text String. For example, if you select Find Whole Words Only and search for "Front Door," FIND does not locate the text string "Front Doormat."

# FLATSHOT

## Quick Reference

Creates a 2D representation of all 3D objects in the current view

**flatshot**
3D Make panel, Flatshot

The Flatshot dialog box (page 592) is displayed.

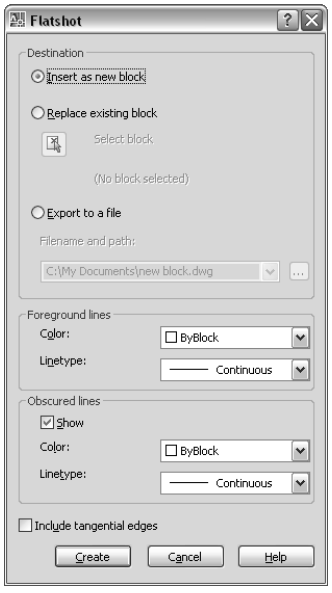
# Flatshot Dialog Box

## Quick Reference

**flatshot**
3D Make panel, Flatshot

Creates a 2D or "flattened" representation of all 3D objects in the current view.

The resulting view is inserted as a block on the XY plane or saved as a file.



### Destination

Controls where the flattened representation is created.

**Insert As New Block** Specifies to insert the flattened representation as a block in the current drawing.

**Replace Existing Block** Replaces an an existing block in the drawing with the newly created block.

**Select Block** Closes the dialog box temporarily while you select the block you are relacing in the drawing. When you finish selecting the block, press ENTER to redisplay the Flatshot dialog box.

**Block Selected** Indicates if a block has been selected.

**Export to a File** Saves the block to an external file.

### Foreground Lines

Contains controls for setting the color and linetype of lines that are not obscured in the flattened view.

**Color** Sets the color of lines that are not obscured in the flattened view.

**Linetype** Sets the linetype of lines that are not obscured in the view.

### Obscured Lines

Controls whether lines that are obscured in the drawing are displayed in the flattened view, and sets the color and linetype of these obscured lines.

**Show** Controls whether obscured lines are shown in the flattened representation. When selected, the 2D flattened representation displays lines hidden by other objects.

**Color** Sets the color of lines that lie behind geometry in the flattened view.

**Linetype** Sets the linetype of lines that lie behind geometry in the flattened view.

**Create** Creates the flattened view.

**Include Tangential Edges** Creates silhouette edges for curved surfaces.

# FOG

## Quick Reference

Obsolete

**fog**

The Render Environment dialog box (page 1200) is displayed.
(*RENDERENVIRONMENT* command)

# FREESPOT

## Quick Reference

Creates a free spotlight, which is similar to a spotlight but without a specified target

**freespot**
Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*

If the LIGHTINGUNITS system variable is set to 0, the following prompt is displayed:

Enter an option to change [Name (page 595)/Intensity (page 595)/Status (page 595)/Hotspot (page 595)/Falloff (page 595)/shadoW (page 596)/Attenuation (page 597)/Color (page 598)/eXit (page 598)] <eXit>:

If the LIGHTINGUNITS system variable is set to 1 or 2, the following prompt is displayed:

Enter an option to change [Name (page 595)/Intensity factor (page 595)/Photometry (page 595)/Status (page 595)/Hotspot (page 595)/Falloff (page 595)/shadoW (page 596)/filterColor (page 598)/eXit (page 598)] <eXit>:

---

**NOTE** When the LIGHTINGUNITS system variable is set to 1 or 2, the Attenuation option has no affect on the creation of the light. It is only maintained for scripting compatibility.

---

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity/Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00 - max float) <1.0000>:

### Hotspot

Specifies the angle that defines the brightest cone of light, which is known to lighting designers as the beam angle. This value can range from 0 to 160 degrees or the equivalent values based on *AUNITS* and *AUNITS*.

Enter hotspot angle (0.00-160.00) <45.0000>:

### Falloff

Specifies the angle that defines the full cone of light, which is also known as the field angle. This value can range from 0 to 160 degrees. The default is 50 degrees or the equivalent values based on *AUNITS* and *AUNITS*. The falloff angle must be greater than or equal to the hotspot angle.

Enter falloff angle (0.00-160.00) <50>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf] <On>:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived

power per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i,** you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*>:

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFtmapped/softsAmpled] <Sharp>:

**Off** Turns off display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory that should be used to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness that should be used to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Disk/Rect] <Disk>:

Specify the sample size by entering **a**.

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v.**

Enter Shape Visibility [Yes/No]<No>:

### Attenuation

Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit]<eXit>:

**Attenuation Type** Controls how light diminishes over distance. The farther away an object is from a spotlight, the darker the object appears. Attenuation is also known as decay.

Enter attenuation type [None/Inverse linear/inverse Squared] <Inverse linear>:

- None. Sets no attenuation. Objects far from the spotlight are as bright as objects close to the light.
- Inverse Linear. Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the spotlight; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.

- Inverse Squared. Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the spotlight; at a distance of 4 units, light is one sixteenth as strong.

**Use Limits** Specifies whether to use limits or not.

Limits [oN/oFf] <Off>:

**Attenuation Start Limit** Specifies the point where light starts as an offset from the center of the light. The default is 0.

Specify start limit offset <1.0000>:

---

**NOTE** Attenuation start limits and end limits are not supported by the OpenGL driver (*wopengl9.hdi*), but the Direct 3D driver (*direct3d9.hdi*) does support end limits for attenuation but does not support start limits. To identify your driver, enter **3dconfig**, and click Manual Tune. Look at the selected Driver Name in the Manual Performance Tuning dialog box.

---

**Attenuation End Limit** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point. Setting an end limit increases performance where the effect of lighting is so minimal that the calculations are wasted processing time.

Specify end limit offset <10.0000>:

### Color/Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]<255,255,255>:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L):

**Color Book** Specifies a color from a color book.

Enter Color Book name:

### Exit

Exits the command.

# FREEWEB

## Quick Reference

Creates a free weblight which is similar to a weblight but without a specified target

**freeweb**
Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*
Enter an option to change [Name (page 599)/Intensity factor (page 599)/Status (page 599)/Photometry (page 599)/weB (page 600)/shadoW (page 601)/filterColor (page 602)/eXit (page 602)] <eXit>:

---

**NOTE** The LIGHTINGUNITS system variable must be set to a value other than 0 to create and use freeweb lights.

---

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00 - max float) <1.0000>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf]:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived power per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i,** you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*> :

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command option.

### Web

Specifies the intensity for a light at points on a spherical grid.

Enter a Web option to change [File/X/Y/Z/Exit] <Exit>:
**File** Enter Web file <>:

Specifies which web file to use to define the properties of the web. Web files have the file extension *.ies*.

**X** Enter Web X rotation <0.0000>:

Specifies the X rotation for the web.

**Y** Enter Web Y rotation <0.0000>:

Specifies the Y rotation for the web.

**Z** Enter Web Z rotation <0.0000>:

Specifies the Z rotation for the web.

**Exit** Exits the command option.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFtmapped/softsAmpled] <Sharp>:
**Off** Turns off display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory to use to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness that should be used to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Linear, Disk, Rect, Sphere, Cyl] <Sphere>:

Specify the sample size by entering **a**.

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v**.

Enter Shape Visibility [Yes/No]<No>:

### Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.
Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.
Enter HSL color (H,S,L) <0,0,100>:

**Color Book** Specifies a color from a color book.
Enter Color Book name:

### Exit

Exits the command.

# G Commands

# 8

**In this chapter**

- GEOGRAPHICLOCATION
- GOTOURL
- GRADIENT
- GRAPHSCR
- GRID
- GROUP

# GEOGRAPHICLOCATION

### Quick Reference

Specifies the latitude and longitude of a location

View ➤ Render ➤ Lights ➤ Geographic Location
**geographiclocation**

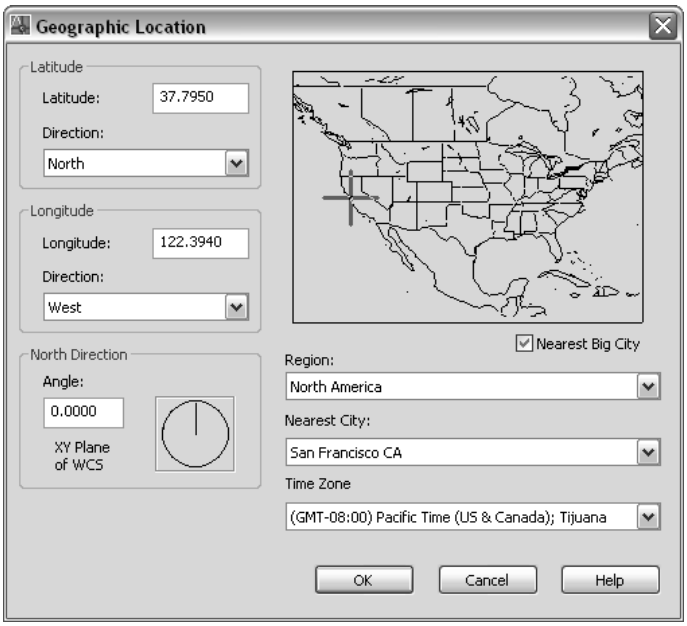
The Geographic Location dialog box (page 604) is displayed.

## Geographic Location Dialog Box

### Quick Reference

Lights 
Light ➤ Geographic Location
View ➤ Render ➤ Lights ➤ Geographic Location
**geographiclocation**

Sets the latitude, longitude, and north direction of a location.




### Latitude

Displays or sets latitude and direction in decimal values.

**Latitude** Sets the latitude of the current location. You can enter a value or select a location on the map. The valid range is 1 to 90. (*LATITUDE* system variable)

**Direction** Sets the direction of the latitude from the equator.

### Longitude

Displays or sets longitude and direction in decimal values. Positive values represent west longitudes.

**Longitude** Displays the longitude of the current location. You can enter a value or select a location on the map. The valid range is 1 to 180. (*LONGITUDE* system variable)

**Direction** Displays the direction of the latitude from the prime meridian.

#### North Direction

Sets the north direction. By default, north is the positive *Y* direction in the world coordinate system (WCS).

**Angle** Specifies the angle from 0 for the north direction.

**North Direction Preview** Displays the north direction.

#### Map

Specifies a location by using the pointing device. The latitude and longitude values are updated when you select a location. If you enter latitude and longitude values, the map updates to show the location.

#### Nearest Big City

Uses the latitude and longitude values for the nearest big city that you select.

#### Region

Specifies the region of the world.

#### Nearest City

Specifies a city in the selected region.

#### Time Zone

Specifies the time zone. Time zones are estimated by reference to the location. You can set the time zone directly. (*TIMEZONE* system variable)

# GOTOURL

## Quick Reference

Opens the file or web page associated with the hyperlink attached to an object

**gotourl**

Select an object that has an attached hyperlink. The file or web page (URL) that is associated with the hyperlink opens.

# GRADIENT

## Quick Reference

Fills an enclosed area or selected objects with a gradient fill

Draw 
Draw ➤ GradientAt the Command prompt, enter gradient.
**gradient**

Opens the Hatch and Gradient dialog box (page 618) to the Gradient tab.

# GRAPHSCR

## Quick Reference

Switches from the text window to the drawing area

**graphscr** (or '**graphscr** for transparent use)

GRAPHSCR closes the text window. You can also press F2 to open or close the text window. This command is ignored on dual-screen systems.

# GRID

## Quick Reference

Displays a grid in the current viewport that is not plotted

Status bar ➤ Grid
**grid** (or '**grid** for transparent use)
Specify grid spacing(X) (page 607) or [ON (page 607)/OFF (page 608)/Snap (page 608)/Major (page 608)/Adaptive (page 608)/Limits (page 608)/Follow (page 608)/Aspect (page 608)] <*current*>: *Specify a value or enter an option*
**Grid Spacing (X)** Sets the grid to the specified value. Entering **x** after the value sets the grid spacing to the specified value multiplied by the snap interval.

**On** Turns on the grid using the current spacing.

GRID turned on

**Off** Turns off the grid.

GRID turned off

**Snap** Sets the grid spacing to the snap interval specified by the *SNAP* command.

**Major** Specifies the frequency of major grid lines compared to minor grid lines. Grid lines rather than grid dots are displayed in any visual style except 2D Wireframe. (*GRIDMAJOR* system variable)

**Adaptive** Controls the density of grid lines when zoomed in or out.

Turn adaptive behavior on [Yes/No] <Yes>: *Enter* **Y** *or* **N**

Limits the density of grid lines or dots when zoomed out. This setting is also controlled by the *GRIDDISPLAY* system variable.

Allow subdivision below grid spacing [Yes/No] <Yes>

If turned on, generates additional, more closely spaced grid lines or dots when zoomed in. The frequency of these grid lines is determined by the frequency of the major grid lines.

**Limits** Displays the grid beyond the area specified by the *LIMITS* command.

**Follow** Changes the grid plane to follow the XY plane of the dynamic UCS. This setting is also controlled by the *GRIDDISPLAY* system variable.

**Aspect** Changes the grid spacing in the *X* and *Y* directions.

Specify the horizontal spacing(X) <*current*>: *Enter a value or press* ENTER

Specify the vertical spacing(Y) <*current*>: *Enter a value or press* ENTER

Entering **x** following either value defines it as a multiple of the snap interval rather than the drawing units.

The Aspect option is not available when the current snap style is Isometric.

GRID set to Aspect

# GROUP

## Quick Reference

Creates and manages saved sets of objects called groups

**group**

The Object Grouping dialog box (page 609)is displayed.

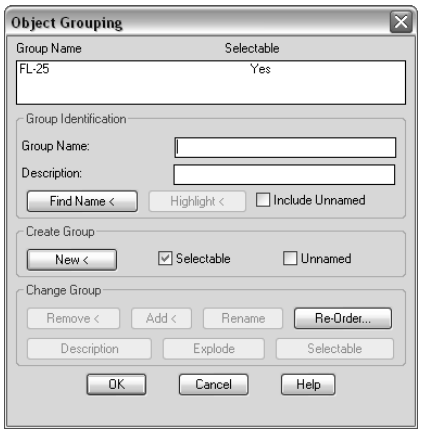
If you enter **-group** at the command prompt, options are displayed at the command prompt (page 614).

# Object Grouping Dialog Box

## Quick Reference

**group**

Displays, identifies, names, and changes object groups.




### Group Name

Displays the names of existing groups.

### Selectable

Specifies whether a group is selectable. When a group is selectable, selecting one object in the group selects the whole group. Objects on locked or frozen layers are not selected. When the PICKSTYLE system variable is set to 0, no groups are selectable.

### Group Identification

Displays the name and description (if any) of the group selected in the Group Name list.

**Group Name** Specifies the group name. Group names can be up to 31 characters long and can include letters, numbers, and the special characters dollar sign ($), hyphen (-), and underscore (_) but not spaces. The name is converted to uppercase characters.

**Description** Displays the description of the selected group, if there is one.

**Find Name** Lists the groups to which an object belongs.

Pick a member of a group: *Select one object*

The Group Member List dialog box is displayed, showing the groups to which the object belongs.

**Highlight** Shows the members of the selected group in the drawing area.

**Include Unnamed** Specifies whether unnamed groups are listed. When this option is cleared, only named groups are displayed.

### Create Group

Specifies properties of new groups.

**New** Creates a new group from the selected objects, using the name and description under Group Name and Description.

desk, computer, and telephone selected

The group name is displayed in the Group Name list.

**Selectable** Specifies that a new group is selectable.

**Unnamed** Indicates that a new group is unnamed. A default name, *A*n*, is assigned to unnamed groups. The *n* represents a number that increases with each new group.

### Change Group

Modifies existing groups.

**Remove** Removes objects from the selected group. To use this option, clear the Selectable option.

Remove objects: *Use an object selection method*

The selected objects are removed from the group. The Object Grouping dialog box is displayed.

telephone selected

telephone removed from group

If you remove all the group's objects, the group remains defined. You can remove the group definition from the drawing by using the Explode option.

**NOTE** When you remove objects from a group and then later add them back during the same drawing session, they are returned to their previous position in the numerical order of the group.

**Add** Adds objects to the selected group.

Select objects: *Use an object selection method*

The selected objects are added to the group. The Object Grouping dialog box is displayed.

chair selected

chair added to group

**NOTE** Group names are displayed in alphabetical order.

**Rename** Renames the selected group to the name entered in Group Name under Group Identification.

**Re-Order** Displays the Order Group dialog box (page 613), in which you can change the numerical order of objects within the selected group. Objects are numbered in the order in which you select them for inclusion in the group. Reordering is useful when creating tool paths. For example, you can change the cut order for the horizontal and vertical lines of a tool path pattern.

You can either change the numerical position of individual group members or ranges of group members or reverse the order of all members. The first object in a group is number 0, not number 1.

**Description** Updates the selected group's description to the name that you enter in Description. You can use up to 64 characters for a description name.

**Explode** Deletes the definition of the selected group. The group's objects remain in the drawing.

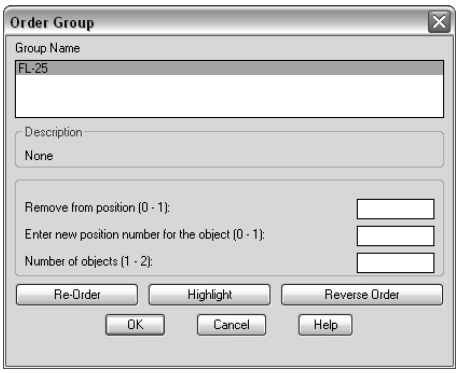
**Selectable** Specifies whether the group is selectable.

# Order Group Dialog Box

## Quick Reference

### group

Reorders objects in groups.




**Group Name** Displays the names of all groups.

**Description** Displays the description of the selected group.

**Remove from Position (0-*n*)** Specifies the current position of the object to reorder.

**Enter New Position Number for the Object (0-*n*)** Specifies the position to which the object moves.

**Number of Objects (1-*n*)** Specifies the object number or range of numbers to reorder.

**Re-Order** Changes the numerical order of objects as specified.

**Highlight** Displays the members of the selected group in the drawing area, one by one, in the current group order.

**Reverse Order** Reverses the order of all group members.

# -GROUP

## Quick Reference

If you enter **-group** at the command prompt, the following GROUP command prompts are displayed.

Enter a group option
[? (page 614)/Order (page 614)/Add (page 614)/Remove (page 615)/Explode (page 615)/REName (page 615)/Selectable (page 615)/Create (page 615)] <Create>: *Enter an option or press* ENTER

### ?—List Groups

Lists names and descriptions of groups defined in the drawing.

Enter group name(s) to list <*>: *Enter a name list or press* ENTER *to list all groups*

### Order

Changes the numerical order of objects within a group. Reordering is useful when creating tool paths. For example, you can change the cut order for the horizontal and vertical lines of a tool path pattern.

Enter a group name or [?]: *Enter a name list or enter* **?** *to list all groups*
Enter position number of the object to reorder (0-*n*) or [Reverse order]: *Enter a position number or enter* **r**
**Position Number** Specifies the position number of the object to reorder. To reorder a range of objects, specify the first object's position number.

Replace at position <0-*n*>: *Enter the position number to which you want the object to move*

Number of objects to re-order <0-*n*>: *Enter the number of objects to reorder*

**Reverse Order** Reverses the order of all members in a group.

### Add

Adds objects to a group.

Enter a group name or [?]: *Enter a name or enter* **?** *to list all groups*
Select objects to add to group...
Select objects:

### Remove

Removes objects from a group.

Enter a group name or [?]: *Enter a name or enter* **?** *to list all groups*
Select objects to remove from group...
Remove objects:

If you remove all the group's objects, the group remains defined. You can remove the group definition from the drawing by using the Explode option.

### Explode

Deletes a group definition by exploding the group into its component objects.

Enter a group name or [?]: *Enter a name or enter* **?** *to list all groups*

### Ungroup

Removes the group name and the association of objects in the group.

### Rename

Assigns a new name to an existing group.

Enter a group name to rename or [?]: *Enter an existing group name or enter* **?** *to list all groups*
Enter a new name for group or [?]: *Enter a new name or enter* **?** *to list all groups*

### Selectable

Specifies whether a group is selectable. When a group is selectable, selecting one object in the group selects the whole group. Objects on locked or frozen layers are not selected.

Enter a group name or [?]: *Enter a name or enter* **?** *to list all groups*
This group is *current*, do you want to change it [Yes/No]? <Y>: *Enter* **y** *or* **n**, *or press* ENTER

### Create

Creates a group.

Enter a group name or [?]: *Enter a name or enter* **?**

Group names can be up to 31 characters long and can include letters, numbers, and special characters dollar sign ($), hyphen (-), and underscore (_) but not spaces. The name is converted to uppercase characters.

Enter a group description: *Enter up to 64 text characters or press* ENTER
Select objects:

# H Commands

# 9

**In this chapter**

- HATCH
- HATCHEDIT
- HELIX
- HELP
- HIDE
- HLSETTINGS
- HYPERLINK
- HYPERLINKOPTIONS

# HATCH

## Quick Reference

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill

Draw 
Draw ➤ HatchAt the Command prompt, enter bhatch.
**hatch**

The Hatch and Gradient dialog box (page 618) is displayed.

If you enter **-hatch** at the command prompt, options are displayed at the command prompt (page 631).

---

**NOTE** By default, the program will not create a hatch pattern that consists of over 10,000 hatch segments. The limit is set by the MaxHatch setting in the registry. To reset the limit to 50,000, for example, enter **(setenv** "**MaxHatch**" "**50000**"**)** at the Command prompt. The limit can be reset to any value between 100 and 10,000,000.

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# Hatch and Gradient Dialog Box

## Quick Reference

Draw
Draw ➤ HatchAt the Command prompt, enter bhatch.
**hatch**

Defines the boundaries, pattern type, pattern properties, and other parameters for hatch and gradient fill objects.

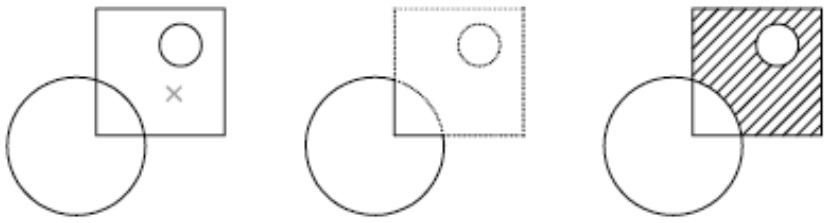
- Hatch tab (page 622)
- Gradient tab (page 625)
- More Options section (page 627)
- Add: Pick Points (page 619)

- Add: Select Objects (page 619)
- Recreate Boundary (page 621)
- Remove Boundaries (page 620)
- View Selections (page 621)
- Options (page 621)
- Inherit Properties (page 622)
- Preview (page 622)

### Add: Pick Points

Determines a boundary from existing objects that form an enclosed area around the specified point. The dialog box closes temporarily, and you are prompted to pick a point.

Pick internal point or [Select objects/remove Boundaries]: *Click within the area to be hatched or filled, specify an option, enter* **u** *or* **undo** *to undo the last selection, or press* ENTER *to return to the dialog box*



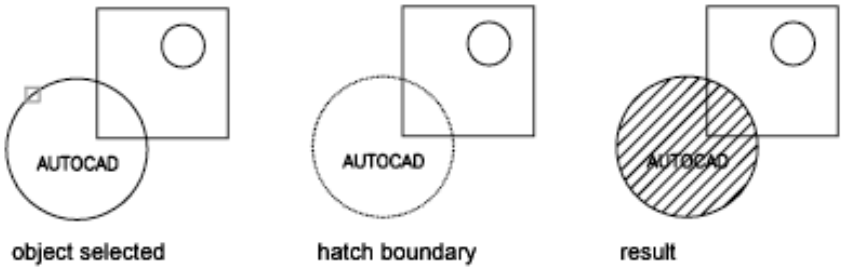

While picking internal points, you can right-click in the drawing area at any time to display a shortcut menu that contains several options.

If you turn on Island Detection, objects that enclose areas within the outermost boundary are detected as islands. How HATCH detects objects using this option depends on which island detection method you select in the More Options area of the dialog box.

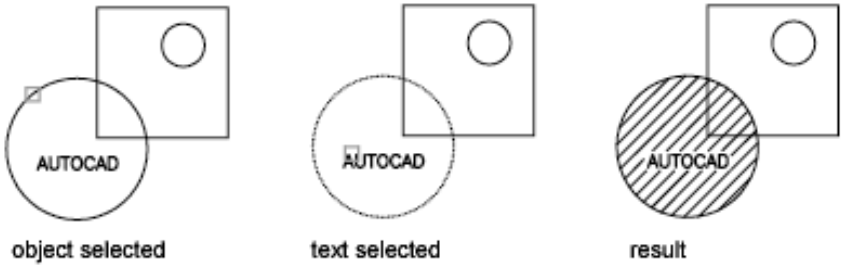
### Add: Select Objects

Determines a boundary from selected objects that form an enclosed area. The dialog box closes temporarily, and you are prompted to select objects.

Select objects or [picK internal point/remove Boundaries]:*Select objects that define the area to be hatched or filled, specify an option, enter* **u** *or* **undo** *to undo the last selection, or press* ENTER *to return to the dialog box*




When you use the Select Objects option, HATCH does not detect interior objects automatically. You must select the objects within the selected boundary to hatch or fill those objects according to the current island detection style.




Each time you click Select Objects, HATCH clears the previous selection set.

While selecting objects, you can right-click at any time in the drawing area to display a shortcut menu. You can undo the last selection or all selections, change the selection method, change the island detection style, or preview the hatch or gradient fill.

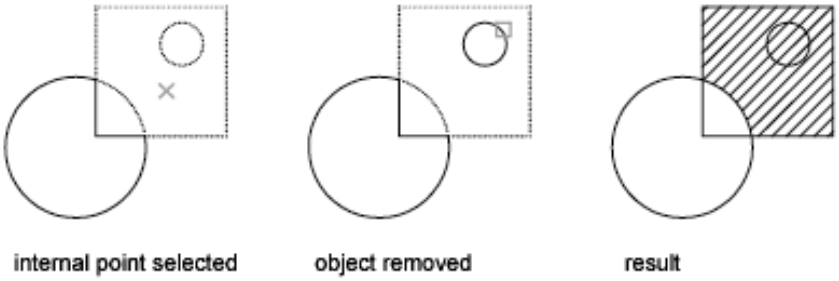
### Remove Boundaries

Removes from the boundary definition any of the objects that were added previously.

When you click Remove Boundaries, the dialog box closes temporarily, and displays a command prompt.

Select Objects or [Add boundaries]: *Select objects to be removed from the boundary definition, specify an option, or press* ENTER *to return to the dialog box*

**Select Objects** Removes temporary boundary objects for hatching or filling as you select them.




**Add Boundaries** Adds temporary boundary objects for hatching or filling as you select them.

### Recreate Boundary

Creates a polyline or region around the selected hatch or fill, and optionally associates the hatch object with it.

When you click Recreate Boundary, the dialog box closes temporarily, and displays a command prompt.

Enter type of boundary object [Region/Polyline] <*current*>: *Enter* **r** *to create a region or* **p** *to create a polyline*
Reassociate hatch with new boundary? [Yes/No] <*current*>: *Enter* **y** *or* **n**

### View Selections

Temporarily closes the dialog box and displays the currently defined boundaries with the current hatch or fill settings. This option is unavailable when no boundary has been defined.

### Options

Controls several commonly used hatch or fill options.

**Annotative** Specifies that the hatch is . Click the information icon to learn more about annotative objects.

**Associative** Controls whether the hatch or fill is associative or nonassociative. A hatch or fill that is associative is updated when you modify its boundaries. (*HPASSOC* system variable)

**Create Separate Hatches** Controls whether a single hatch object or multiple hatch objects are created when several separate closed boundaries are specified. (*HPSEPARATE* system variable)

**Draw Order** Assigns the draw order to a hatch or fill. You can place a hatch or fill behind all other objects, in front of all other objects, behind the hatch boundary, or in front of the hatch boundary. (*HPDRAWORDER* system variable)

#### Inherit Properties

Hatches or fills specified boundaries using the hatch or fill properties of a selected hatch object. *HPINHERIT* controls whether the hatch origin of the resulting hatch is determined by *HPORIGIN* or by the source object. After selecting the hatch object whose properties you want the hatch to inherit, you can right-click in the drawing area and use the options on the shortcut menu to switch between the Select Objects and Pick Internal Point options to create boundaries.

When you click Inherit Properties, the dialog box closes temporarily, and displays a command prompt.

Select hatch object: *Click within a hatched or filled area to select the hatch whose properties are to be used for the new hatch object*

#### Preview

Dismisses the dialog box and displays the currently defined boundaries with the current hatch or fill settings. Click in the drawing or press ESC to return to the dialog box. Right-click or press ENTER to accept the hatch or fill. This option is not available when you have not yet specified points or selected objects to define your boundaries.

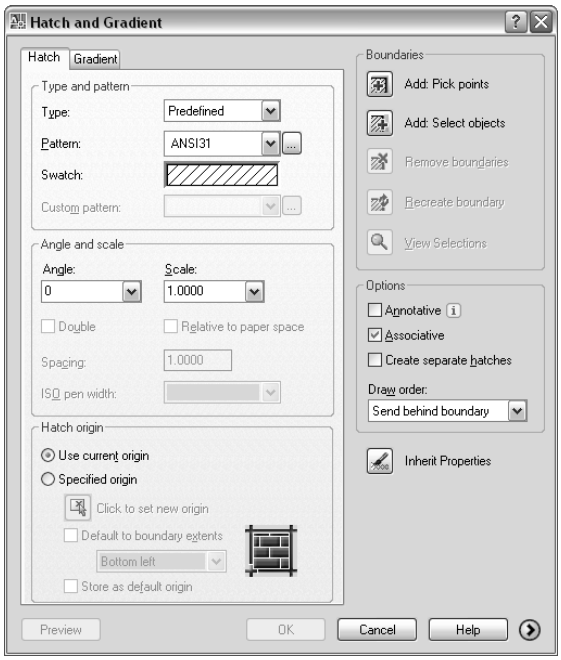
#### More Options

Expands the dialog box to display more options.

## Hatch Tab (Hatch and Gradient Dialog Box)

### Quick Reference

Defines the appearance of the hatch pattern to be applied.




### Type and Pattern

Specifies the type of hatch and the pattern.

**Type** Sets the pattern type. User-defined patterns are based on the current linetype in your drawing. A custom pattern is a pattern that is defined in any custom PAT files that you have added to the search path. You can control the angle and scale of any pattern.

Predefined patterns are stored in the *acad.pat* or *acadiso.pat* files supplied with the product.

**Pattern** Lists the available predefined patterns. The six most recently used predefined patterns appear at the top of the list. HATCH stores the selected pattern in the *HPNAME* system variable. The Pattern option is available only if you set Type to Predefined.

**[...] Button** Displays the Hatch Pattern Palette dialog box (page 630), in which you can view preview images for all predefined patterns at once to help you make a selection.

**Swatch** Displays a preview of the selected pattern. You can click the swatch to display the Hatch Pattern Palette dialog box (page 630). When the SOLID

pattern is selected, you can click the right arrow to display a list of colors or the Select Color dialog box (page 251).

**Custom Pattern** Lists the available custom patterns. The six most recently used custom patterns appear at the top of the list. The name of the selected pattern is stored in the *HPNAME* system variable. The Custom Pattern option is available only if you set Type to Custom.

**[...] Button** Displays the Hatch Pattern Palette dialog box (page 630), in which you can view preview images for all custom patterns at once to help you make a selection.

### Angle and Scale

Specifies an angle and a scale for the selected hatch pattern.

**Angle** Specifies an angle for the hatch pattern relative to the *X* axis of the current UCS. HATCH stores the angle in the *HPANG* system variable.

**Scale** Expands or contracts a predefined or custom pattern. HATCH stores the scale in the *HPSCALE* system variable. This option is available only if you set Type to Predefined or Custom.

**Double** For user-defined patterns, draws a second set of lines positioned at 90 degrees to the original lines, creating a crosshatch. This option is available only if you set Type to User Defined on the Hatch tab. (*HPDOUBLE* system variable)

**Relative to Paper Space** Scales the hatch pattern relative to paper space units. Using this option, you can easily display hatch patterns at a scale that is appropriate for your layout. This option is available only from a layout.

**Spacing** Specifies the spacing of lines in a user-defined pattern. HATCH stores the spacing in the *HPSPACE* system variable. This option is available only if you set Type to User Defined.

**ISO Pen Width** Scales an ISO predefined pattern based on the selected pen width. This option is available only if you set Type to Predefined and set Pattern to one of the available ISO patterns.

### Hatch Origin

Controls the starting location of hatch pattern generation. Some hatches, such as brick patterns, need to be aligned with a point on the hatch boundary. By default, all hatch origins correspond to the current UCS origin.

**Use Current Origin** Uses the setting stored in the *HPORIGINMODE* system variable. The origin is set to 0,0 by default.

**Specified Origin** Specifies a new hatch origin. Click this option to make the following options available.

**Click to Set New Origin** Specifies the new hatch origin point directly.

**Default to Boundary Extents** Calculates a new origin based on the rectangular extents of the boundary for the hatch object. Choices include each of the four corners of the extents and its center. (*HPORIGINMODE* system variable)

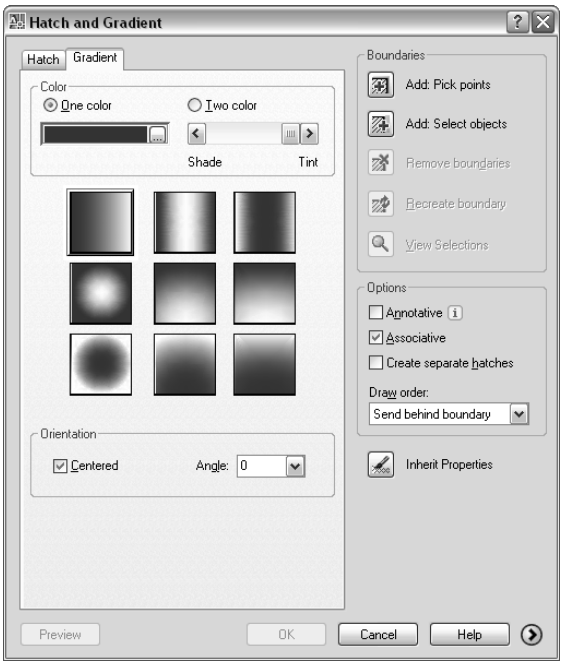
**Store as Default Origin** Stores the value of the new hatch origin in the *HPORIGIN* system variable.

**Origin Preview** Shows the current location of the origin.

## Gradient Tab (Hatch and Gradient Dialog Box)

### Quick Reference

Defines the appearance of the gradient fill to be applied.




### Color

**One Color** Specifies a fill that uses a smooth transition between darker shades and lighter tints of one color. When One Color is selected, HATCH displays a color swatch with Browse button and a Shade and Tint slider.

**Two Color** Specifies a fill that uses a smooth transition between two colors. When Two Color is selected, HATCH displays a color swatch with a Browse button for color 1 and for color 2.

**Color Swatch** Specifies the color for the gradient fill. Click the Browse button [...] to display the Select Color dialog box (page 251), where you can select an AutoCAD Color Index (ACI) color, true color, or color book color. The default color displayed is the current color in the drawing.

**Shade and Tint Slider** Specifies the tint (the selected color mixed with white) or shade (the selected color mixed with black) of a color to be used for a gradient fill of one color.

### Gradient Patterns

Displays nine fixed patterns for gradient fills. These patterns include linear sweep, spherical, and parabolic.

### Orientation

Specifies the angle of the gradient and whether it is symmetrical.

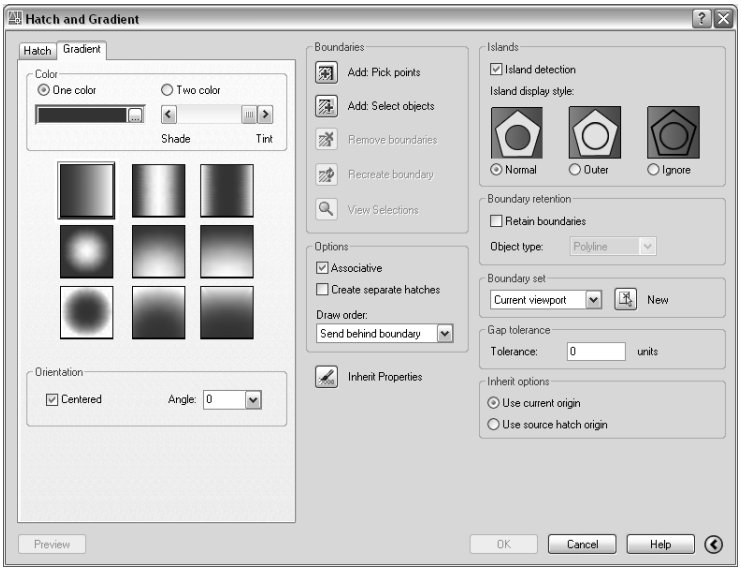
**Centered** Specifies a gradient configuration that is symmetrical. If this option is not selected, the gradient fill is shifted up and to the left, creating the illusion of a light source to the left of the object.

**Angle** Specifies the angle of the gradient fill. The specified angle is relative to the current UCS. This option is independent of the angle specified for hatch patterns.

## More Options (Hatch and Gradient Dialog Box)

### Quick Reference

Controls the operation of islands and boundaries.



### Islands

Specifies the method used to hatch or fill objects within the outermost boundary. If no internal boundaries exist, specifying an island detection style has no effect. Because you can define a precise set of boundaries, it is often best to use the Normal style.

**Island Detection** Controls whether internal closed boundaries, called islands, are detected.

**Normal** Hatches or fills inward from the outer boundary. If HATCH encounters an internal island, it turns off hatching or filling until it encounters another island within the island. You can also set the Normal style by adding **,N** to the pattern name in the *HPNAME* system variable.

**Outer** Hatches or fills inward from the outer boundary. HATCH turns hatching or filling off if it encounters an internal island. This option hatches or fills only the outermost level of the structure and leaves the internal structure blank. You can also set the Outer style by adding **,O** to the pattern name in the *HPNAME* system variable.

**Ignore** Ignores all internal objects and hatches or fills through them. You can also set the Ignore style by adding **,I** to the pattern name in the *HPNAME* system variable.

The Normal, Outer, and Ignore options are also available from a shortcut menu by right-clicking in the drawing area while you specify points or select objects to define your boundaries.

### Boundary Retention

Specifies whether to retain boundaries as objects, and determines the object type applied to those objects.

**Retain Boundaries** Creates boundary objects from the temporary hatch boundaries and adds them to the drawing.

**Object Type** Controls the type of the new boundary object. The resulting boundary object can be a region or a polyline object. This option is available only if Retain Boundaries is checked.

For more information about regions, see "Create and Combine Areas (Regions)" in the *User's Guide*.

### Boundary Set

Defines the set of objects analyzed when defining a boundary from a specified point. The selected boundary set has no effect when you use Select Objects to define a boundary.

By default, when you use the Add: Pick Point option to define a boundary, HATCH analyzes all objects in the current viewport extents. By redefining the boundary set, you can disregard certain objects when defining boundaries without having to hide or remove those objects. For large drawings, redefining the boundary set can also produce the boundary faster because HATCH examines fewer objects.

**Current Viewport** Defines the boundary set from everything in the current viewport extents. Selecting this option discards any current boundary set.

**Existing Set** Defines the boundary set from the objects that you selected with New. If you have not created a boundary set with New, the Existing Set option is not available.

**New** Prompts you to select the objects that define the boundary set.

### Gap Tolerance

Sets the maximum size of gaps that can be ignored when objects are used as a hatch boundary. The default value, 0, specifies that the objects must enclose the area with no gaps.

Enter a value, in drawing units, from 0 to 5000 to set the maximum size of gaps that can be ignored when the objects serve as a hatch boundary. Any gaps equal to or smaller than the value you specify are ignored, and the boundary is treated as closed. (*HPGAPTOL* system variable)

### Inherit Options

When you create a hatch with Inherit Properties, these settings control the location of the hatch origin.

**Use Current Origin** Uses the current hatch origin setting

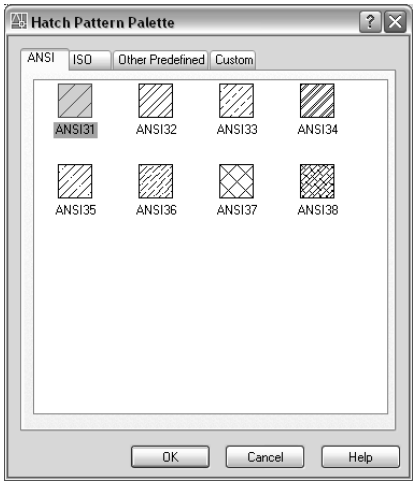
**Use Source Hatch Origin** Uses the hatch origin of the source hatch

# Hatch Pattern Palette Dialog Box

## Quick Reference

Draw
Draw ➤ HatchAt the Command prompt, enter bhatch.
**hatch**

Displays preview images for all predefined and custom patterns. The dialog box organizes patterns on four tabs with images arranged alphabetically on each tab. Click an image to select a pattern and click OK.




**ANSI** Displays all ANSI patterns shipped with the product.

**ISO** Displays all ISO patterns shipped with the product.

**Other Predefined** Displays all patterns other than ANSI and ISO shipped with the product.

**Custom** Displays a list of custom PAT files that you have added to the search path, which is set in the Options dialog box, Files tab.

**Preview** Displays a preview image of the selected custom pattern.

# -HATCH

## Quick Reference

If you enter **-hatch** at the command prompt, the following HATCH command prompts are displayed.

Specify internal point (page 631) or [Properties (page 631)/Select objects (page 632)/draW boundary (page 632)/remove Boundaries (page 632)/Advanced (page 633)/DRaw order (page 637)/Origin (page 637)/ANnotative (page 637)]: *Specify a point or enter an option*

### Internal Point

Determines a boundary from existing objects that form an enclosed area around the specified point. If you turn on Island Detection, objects that enclose areas within the outermost boundary are detected as islands. How HATCH detects objects using this option depends on which island detection method is specified.

### Properties

Specifies new hatch pattern properties to apply.

Enter a pattern name or [?/Solid/User defined] <*current*>: *Enter a predefined or custom pattern name, enter* **s**, *enter* **u**, *enter* **?**, *or press* ENTER
**Pattern Name—Predefined or Custom** Specifies a predefined pattern in the *acad.pat* or *acadiso.pat* file or a custom pattern in its own PAT file.

Enter the pattern name followed by an optional hatch style code. Precede the pattern name with an asterisk (*) to fill the area with individual lines instead of a hatch object.

Specify a scale for the pattern <*current*>: *Specify a scale or press* ENTER

Specify an angle for the pattern <*current*>: *Specify an angle or press* ENTER

**?—List Pattern Names** Lists and describes the hatch patterns defined in the *acad.pat* file.

Pattern(s) to list <*>: *Enter a name list or press* ENTER

**Solid** Specifies a solid fill and redisplays the first HATCH command line prompt, where you can define a boundary.

**User Defined** Specifies a user-defined pattern. Enter **u**, followed by an optional hatch style code. Precede the **u** with an asterisk (*****) to fill the area with individual lines instead of a hatch block.

Specify angle for crosshatch lines <*current*>: *Specify an angle for the pattern or press* ENTER

Specify spacing between the lines <*current*>: *Specify the distance between pattern lines or press* ENTER

Double hatch area? [Yes/No] <*current*>: *Enter* **y** *to specify a second set of lines to be drawn at 90 degrees to the original lines, or press* ENTER

### Select Objects

Determines a boundary from selected objects that form an enclosed area. Adds objects to the boundary definition.

Select objects: *Use an object selection method*

### Draw Boundary

Uses specified points to define the boundary of a hatch or fill.

Retain polyline boundary? [Yes/No] <*current*>: *Enter* **y** *to retain the polyline hatch boundary object or* **n** *to discard it after the area is hatched*
Specify start point: *Specify the start point of the polyline boundary*
Specify next point or [Arc/Close/Length/Undo]: *Specify a point, enter an option, or press* ENTER

These options are also available in the *PLINE* command. When you complete the polyline boundary, press ENTER. You can then create additional polyline boundaries, or press ENTER to twice to apply the hatch.

Specify start point for new boundary or <apply hatch>: *Specify a point or press* ENTER *twice to apply the hatch*

### Remove Boundaries

Removes from the boundary definition any of the objects that were added previously.

Select Objects or [Add boundaries]: *Select objects to be removed from the boundary definition, enter an option, or press* ENTER *to return to the previous prompt*
**Select Objects** Removes objects from the boundary definition.

**Add Boundaries** Adds objects to the boundary definition.

### Advanced

Sets the method used to create the hatch boundary.

Enter an option [Boundary set/Retain boundary/Island detection/Style/Associativity/Gap tolerance/separate Hatches]: *Enter an option or press* ENTER *to return to the previous prompt*

### Boundary Set

Defines the set of objects to be analyzed when defining a boundary from a specified internal point.

Specify candidate set for boundary [New/Everything] <*current*>: *Enter an option or press* ENTER
**New** Creates a boundary set from a selection set you define.

Select objects: *Use an object selection method*

**Everything** Creates a boundary set from everything visible in the current viewport. Selecting this option discards any current boundary set and uses everything visible in the drawing or in the current viewport.

### Retain Boundary

Specifies whether to add the temporary boundary objects to the drawing after hatching is completed.

Retain derived boundaries? [Yes/No] <*current*>: *Enter* **y** *or* **n***, or press* ENTER

### Island Detection

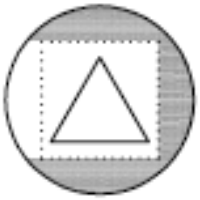
Specifies whether to use objects within the outermost boundary as boundary objects.

Do you want island detection? [Yes/No] <*current*>: *Enter* **y** *or* **n***, or press* ENTER

Specifying no island detection prompts for the ray casting method.

Enter type of ray casting [Nearest/+X/-X/+Y/-Y/Angle] <*current*>: *Enter an option or press* ENTER
**Nearest** Runs a line from the point you specify to the nearest object and then traces the boundary in a counterclockwise direction.

**+X** Runs a line in the positive *X* direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.




**-X** Runs a line in the negative *X* direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.


**+Y** Runs a line in the positive *Y* direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.


**-Y** Runs a line in the negative *Y* direction from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.


**Angle** Runs a line at the specified angle from the point you specify to the first object encountered and then traces the boundary in a counterclockwise direction.

Specify ray casting angle: *Specify an angle*

### Style

Specifies the method used to hatch objects within the outermost hatch boundary. If you have selected no internal objects, a hatching style has no effect. Because you can define a precise set of boundaries, it's often best to use the Normal style.

Enter hatching style [Ignore/Outer/Normal] <*current*>: *Enter an option or press* ENTER

**Ignore** IIgnores all internal objects and hatches or fills through them. You can also set the Ignore style by adding **,I** to the pattern name in the *HPNAME* system variable.

**Outer** Hatches or fills inward from the outer boundary. HATCH turns hatching or filling off if it encounters an internal island. This option hatches or fills only the outermost level of the structure and leaves the internal structure blank. You can also set the Outer style by adding **,O** to the pattern name in the *HPNAME* system variable.

**Normal** Hatches or fills inward from the outer boundary. If HATCH encounters an internal island, it turns off hatching or filling until it encounters another island within the island. You can also set the Normal style by adding **,N** to the pattern name in the *HPNAME* system variable.

### Associativity

Specifies that the new hatch pattern is updated when its boundaries are modified.

A new hatch pattern's associativity is set by the *HPASSOC* system variable. Changing the associativity setting here also changes the setting in the HPASSOC system variable.

Do you want associativity? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

### Gap Tolerance

Sets the maximum size of gaps that can be ignored when objects are used as a hatch boundary. The default value, 0, specifies that the objects must enclose the area with no gaps.

Specify a boundary gap tolerance value <0>: *Enter a value, in drawing units, from 0 to 5000*

Any gaps equal to or smaller than the value you specify in the gap tolerance are ignored, and the boundary is treated as closed.

### Separate Hatches

Controls whether the command creates a single hatch object or multiple hatch objects when several separate closed boundaries are specified.

Create separate hatches? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

### Draw Order

Assigns the draw order to a hatch or fill. You can place a hatch or fill behind all other objects, in front of all other objects, behind the hatch boundary, or in front of the hatch boundary. (*HPDRAWORDER* system variable)

Enter draw order [do Not assign/send to Back/bring to Front/send beHind boundary/bring in front of bounDary] <send beHind boundary>: *Enter an option, or press* ENTER *to accept the default and return to the previous prompt*

### Origin

Controls the starting location of hatch pattern generation. Some hatches, such as brick patterns, need to be aligned with a point on the hatch boundary. By default, all hatch origins correspond to the current UCS origin.

[Use current origin/Set new origin/Default to boundary extents] <*current*>: *Enter an option, or press* ENTER *to accept the default and return to the previous prompt*
**Use current origin** Sets the value of the HPORIGINMODE system variable. The last 5 options listed below correspond to the values 1-5.

[Use current origin/Set new origin/bottom Left/bottom Right/top rIght/top lEft/Center] <*current*>: *Enter an option, or press* ENTER *to accept the default and return to the previous prompt*

**Set new origin** Specifies the new hatch origin point directly.

**Default to boundary extents** Calculates a new origin based on the rectangular extents of the hatch. Choices include each of the four corners of the extents and its center. Also can store the value of the new hatch origin in the HPORIGIN system variable.

### Annotative

Specifies that the hatch is .

Create annotative hatch [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

# HATCHEDIT

### Quick Reference

Modifies an existing hatch or fill

Modify II 
Modify ➤ Object ➤ HatchAt the Command prompt, enter hatchedit.
Select a hatch object to edit, and right-click in the drawing area. Click Hatch Edit.
**hatchedit**
Select hatch object: *Use an object selection method*

The Hatch Edit dialog box (page 638) is displayed.

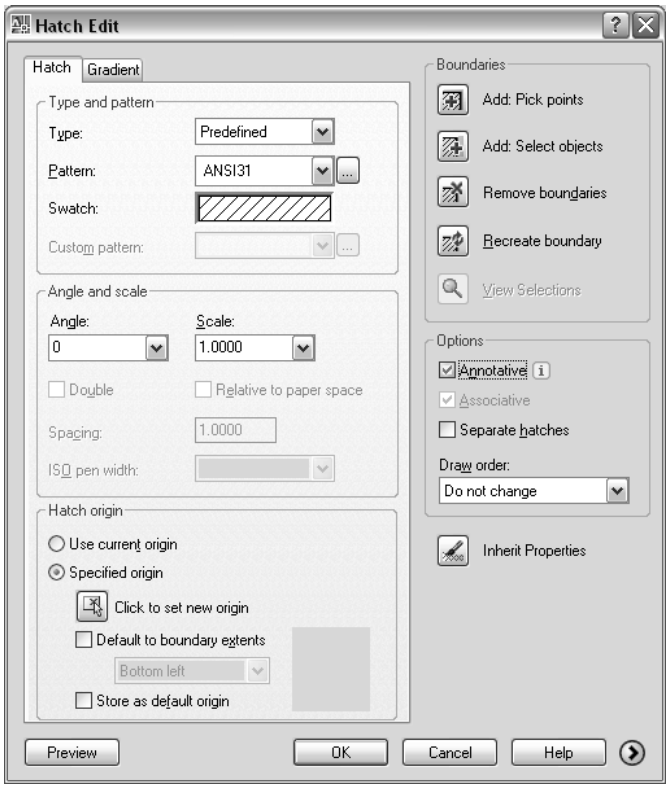
If you enter **-hatchedit** at the command prompt, options are displayed at the command prompt (page 640).

## Hatch Edit Dialog Box

### Quick Reference

Modify II
Modify ➤ Object ➤ HatchAt the Command prompt, enter hatchedit.
Select a hatch object to edit, right-click in the drawing area, and choose Hatch Edit.
**hatchedit**

Modifies the characteristics of an existing hatch or fill. The Hatch Edit dialog box displays the current properties of the selected hatch or fill object. You can modify only the properties in the Hatch Edit dialog box that are available.




**Hatch Tab** Modifies the hatch pattern properties.

For information about these options, see the Hatch tab (page 622) in the Hatch and Gradient dialog box.

**Gradient Tab** Modifies the gradient fill properties. For information about these options, see the Gradient tab (page 625) in the Hatch and Gradient dialog box.

**More Options** Modifies the Inherit Options setting that is associated with Inherit Properties. All other options are not available.

For more information, see Inherit Properties in the Hatch and Gradient dialog box (page 618).

**Boundaries** Modifies the boundaries of a hatch or fill.

For more information, see the Hatch and Gradient dialog box (page 618).

**Options** Makes hatch , removes hatch associativity, separates a single hatch object that has several separate boundaries into individual hatch objects, and changes hatch draw order.

For more information, see the Hatch and Gradient dialog box (page 618).

**Inherit Properties** Assigns the hatch or fill properties from another hatch object.

For more information, see the Hatch and Gradient dialog box (page 618).

**Preview** Temporarily dismisses the dialog box and displays the selected object with the specified properties.

# -HATCHEDIT

## Quick Reference

If you enter **-hatchedit** at the command prompt, the following HATCHEDIT command prompts are displayed.

Select hatch object:
Enter hatch option [DIssassociate (page 640)/Style (page 640)/Properties (page 641)/DRaw order (page 641)/ADd boundaries (page 641)/Remove boundaries (page 641)/recreate Boundary (page 641)/Associate (page 642)/separate Hatches (page 642)/ANnotative (page 642)] <Properties>: *Enter an option or press* ENTER

### Disassociate

Removes the associative quality from an associative hatch.

### Style

Changes the hatch style type. The following prompt is displayed:

Enter hatching style [Ignore/Outer/Normal] <*current*>:
**Ignore** Ignores all internal objects and hatches or fills through them. You can also set the Ignore style by adding **,I** to the pattern name in the *HPNAME* system variable.

**Outer** Hatches or fills inward from the outer boundary. HATCH turns hatching or filling off if it encounters an internal island. This option hatches or fills only the outermost level of the structure and leaves the internal structure blank. You can also set the Outer style by adding **,O** to the pattern name in the *HPNAME* system variable.

**Normal** Hatches or fills inward from the outer boundary. If HATCH encounters an internal island, it turns off hatching or filling until it encounters another island within the island. You can also set the Normal style by adding **,N** to the pattern name in the *HPNAME* system variable.

### Properties

Specifies new hatch properties for the selected hatch. For an explanation of setting pattern properties at the command prompt, see *HATCH*.

### Draw Order

Sets the draw order of the hatch. The following prompt is displayed:

Enter draw order [do Not change/send to Back/bring to Front/send beHind boundary/bring in front of bounDary] <do Not change>: *Enter an option*
**Send to Back** Sends the hatch in back of all other objects

**Bring to Front** Brings the hatch in front of all other objects

**Send Behind Boundary** Sends the hatch behind the hatch boundary

**Bring in Front of Boundary** Brings the hatch in front of the hatch boundary

### Add Boundaries

Modifies the boundaries of a hatch or fill by adding boundaries.

For more information, see Add: Pick Points (page 619) or Add: Select Objects (page 619).

### Remove Boundaries

Modifies the boundaries of a hatch or fill by removing boundaries.

For more information, see Remove Boundaries (page 620).

### Recreate Boundary

Creates a polyline or region around the selected hatch or fill, and optionally associates the hatch object with it.

For more information, see Recreate Boundary (page 621).

### Associate

Specifies that the selected hatch pattern be associated with the specified boundary objects.

Specify internal point or [Select Objects]: *Click within a bounded area, enter an option, or press* ENTER *to return to the previous prompt*
**Specify Internal Point** Determines a boundary from existing objects that form an enclosed area around the specified point. Associates the selected hatch with that boundary.

**Select Objects** Determines a boundary from selected objects that form an enclosed area. Associates the selected hatch with that boundary.

### Separate Hatches

Separates a single hatch object that has several separate boundaries into individual hatch objects.

### Origin

Controls the starting location of hatch pattern generation. Some hatches, such as brick patterns, need to be aligned with a point on the hatch boundary. By default, all hatch origins correspond to the current UCS origin.

[Use current origin/Set new origin/Default to boundary extents] <*current*>: *Enter an option, or press* ENTER *to accept the default and return to the previous prompt*
**Use current origin** Sets the value of the HPORIGINMODE system variable. The last 5 options listed below correspond to the values 1-5.

[Use current origin/Set new origin/bottom Left/bottom Right/top rIght/top lEft/Center] <*current*>: *Enter an option, or press* ENTER *to accept the default and return to the previous prompt*

**Set new origin** Specifies the new hatch origin point directly.

**Default to boundary extents** Calculates a new origin based on the rectangular extents of the hatch. Choices include each of the four corners of the extents and its center. Also can store the value of the new hatch origin in the HPORIGIN system variable.

### Annotative

Specifies that the hatch is .

Create annotative hatch [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

# HELIX

## Quick Reference

Creates a 2D or 3D spiral

Modeling 
Draw ➤ HelixAt the Command prompt, enter Helix.
**helix**
3D Make panel (click icon to expand), Helix
Number of turns = 3 *(default)*
Twist = CCW *(default)*
Specify center point of base: *Specify a point*
Specify base radius or [Diameter (page 643)] <1.0000>: *Specify a base radius, enter* **d** *to specify the diameter, or press* ENTER *to specify the default base radius value*
Specify top radius or [Diameter (page 643)] <1.0000>: *Specify a top radius, enter* **d** *to specify the diameter, or press* ENTER *to specify the default top radius value*
Specify helix height or [Axis endpoint (page 644)/Turns (page 644)/turn Height (page 644)/tWist (page 644)] <1.0000>: *Specify a helix height, or enter an option*

Initially, the default base radius is set to 1. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive or helix.

The default value for the top radius is always the value of the base radius.

The base radius and top radius cannot both be set to 0.

### Diameter (Base)

Specifies the diameter of the base of the helix.

Specify diameter <2.0000>: *Specify a diameter or press* ENTER *to specify the default value*

Initially, the default base diameter is set to 2. During a drawing session, the default value for the base diameter is always the previously entered base diameter value.

### Diameter (Top)

Specifies the diameter of the top of the helix.

Specify diameter <2.0000>: *Specify a diameter or press* ENTER *to specify the default value*

The default value for the top diameter is always the value of the base diameter.

### Axis Endpoint

Specifies the endpoint location for the helix axis. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the helix.

Specify axis endpoint: *Specify a point*

### Turns

Specifies the number of turns (revolutions) for the helix. The number of turns for a helix cannot exceed 500.

Initially, the default value for the number of turns is three. During a drawing session, the default value for the number of turns is always the previously entered number of turns value.

Enter number of turns: *Enter a number*

### Turn Height

Specifies the height of one complete turn within the helix.

The number of turns in the helix will automatically update accordingly when a turn height value is specified. If the number of turns for the helix has been specified, you cannot enter a value for the turn height.

Specify distance between turns <default>: *Enter a number to specify the height for each turn in the helix*

### Twist

Specifies whether the helix is drawn in the clockwise (CW) or the counterclockwise (CCW) direction. The default value for the helix twist is CCW.

Enter twist direction of helix [CW/CCW] <CCW>: *Specify a twist direction for the helix*

# HELP

## Quick Reference

Displays Help

Standard 
Help ➤ HelpAt the Command prompt, enter help.
**help** or **?** or press F1 (or '**help** for transparent use)

To display Help for a menu, open the menu, and then press F1.

Pressing F1 or entering '**help** while a command is active displays Help for that command. Choosing the Help button in a dialog box displays Help for that dialog box.

# HIDE

## Quick Reference

Regenerates a three-dimensional wireframe model with hidden lines suppressed

Render 
View ➤ HideAt the Command prompt, enter hide.
**hide**

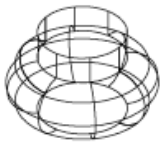
When you use *VPOINT*, *DVIEW*, or *VIEW* to create a 3D view of your 2D drawing, a wireframe is displayed in the current viewport. All lines are present, including those hidden by other objects. HIDE eliminates the hidden lines from the screen.

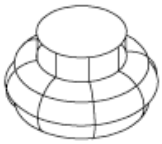
In a 3D drawing, HIDE starts *VSCURRENT* and sets the visual style to 3D Hidden in the current viewport. You can view all the settings for 3D Hidden in the Visual Styles Manager (page 1520).

HIDE considers the following to be opaque surfaces that hide objects: circles, solids, traces, text, regions, wide polyline segments, 3D faces, polygon meshes, and the extruded edges of objects with nonzero thickness.

If they are extruded, circles, solids, traces, and wide polyline segments are treated as solid objects with top and bottom faces. You cannot use HIDE on objects whose layers have been frozen; however, you can use HIDE on objects whose layers have been turned off.

In order to hide text created with DTEXT, MTEXT, or TEXT, the *HIDETEXT* system variable must be set to 1 or the text must be assigned a thickness value.




When using the HIDE command, if the *INTERSECTIONDISPLAY* system variable is on, face-to-face intersections of 3D surfaces are displayed as polylines.

The 3D Hidden visual style does not honor the setting of INTERSECTIONDISPLAY.

If the *DISPSILH* system variable is on, HIDE displays 3D solid objects with silhouette edges only. It won't show the internal edges produced by objects that have facets.

If the HIDETEXT system variable is off, HIDE ignores text objects when producing the hidden view. Text objects are always displayed regardless of whether they are obscured by other objects, and objects obscured by text objects are unaffected.

# HLSETTINGS

## Quick Reference

Controls the display properties of models

**hlsettings**

The Visual Styles Manager (page 1520) is displayed.

# HYPERLINK

## Quick Reference

Attaches a hyperlink to an object or modifies an existing hyperlink

Insert ➤ HyperlinkAt the Command prompt, enter hyperlink.
To edit a hyperlink, select an object that contains a hyperlink, right-click in the drawing area, and choose Hyperlink ➤ Edit Hyperlink.
**hyperlink**
Select objects: *Use an object selection method*

One of the following dialog boxes is displayed:

- Insert Hyperlink dialog box (for graphical objects that do not already contain hyperlinks) (page 647)
- Edit Hyperlink dialog box (for graphical objects that already contain hyperlinks) (page 650)

After you insert a hyperlink, the hyperlink icon is displayed when you move the cursor over the attached object. To open the hyperlink, right-click the selected object and choose Hyperlink. The *PICKFIRST* system variable must be set to 1 to open files associated with hyperlinks.

If you enter **-hyperlink** at the command prompt, options are displayed at the command prompt (page 651), including an option to define an area to associate with a hyperlink.

# Insert Hyperlink Dialog Box

## Quick Reference

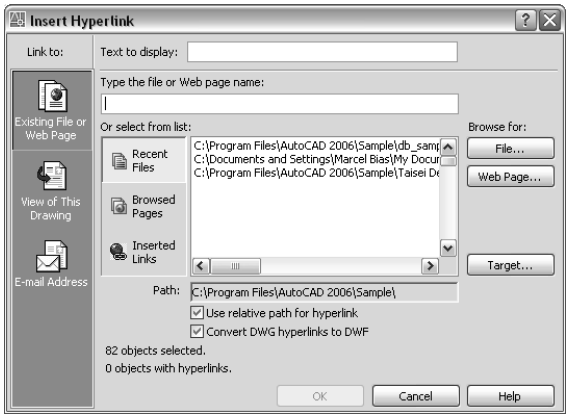
Insert ➤ HyperlinkAt the Command prompt, enter hyperlink.
To edit a hyperlink, select an object that contains a hyperlink, right-click in the drawing area, and choose Hyperlink ➤ Edit Hyperlink.
**hyperlink**

Attaches hyperlinks to graphical objects.

**Text to Display** Specifies a description for the hyperlink. This is useful when the file name or URL is not helpful in identifying the contents of the linked file.

### Existing File or Web Page Tab (Insert Hyperlink Dialog Box)

Creates a hyperlink to an existing file or web page.




**Type the File or Web Page Name** Specifies the file or web page to associate with the hyperlink. The file can be stored locally, on a network drive, or on an Internet or intranet location.

**Recent Files** Displays a list of recently linked files, from which you can select one to link to.

**Browsed Pages** Displays a list of recently browsed web pages, from which you can select one to link to.

**Inserted Links** Displays a list of recently inserted hyperlinks, from which you can select one to link to.

**File** Opens the Browse the Web - Select Hyperlink dialog box (a standard file selection dialog box), in which you can navigate to the file that you want to associate with the hyperlink.

**Web Page** Opens the browser, in which you can navigate to a web page that you want to associate with the hyperlink.

**Target** Opens the Select Place in Document dialog box (page 650), in which you can select a named location in a drawing to link to.

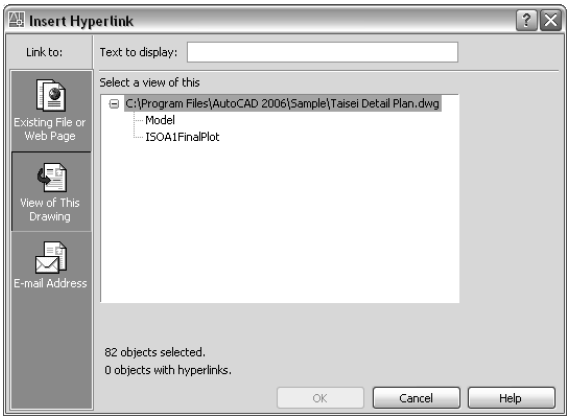
**Path** Displays the path to the file associated with the hyperlink. If Use Relative Path for Hyperlink is selected, only the file name is listed. If Use Relative Path for Hyperlink is cleared, the full path and the file name are listed.

**Use Relative Path for Hyperlink** Sets a relative path for hyperlinks. If you select this option, the full path to the linked file is not stored with the hyperlink. The relative path is set to the value specified by the *HYPERLINKBASE* system variable or, if no value is specified for HYPERLINKBASE, to the current drawing path. If this option is cleared, the full path to the associated file is stored with the hyperlink.

**Convert DWG Hyperlinks to DWF** Specifies that the DWG hyperlink will convert to a DWF file hyperlink when you publish or plot the drawing to a DWF file.

### View of This Drawing Tab (Insert Hyperlink Dialog Box)

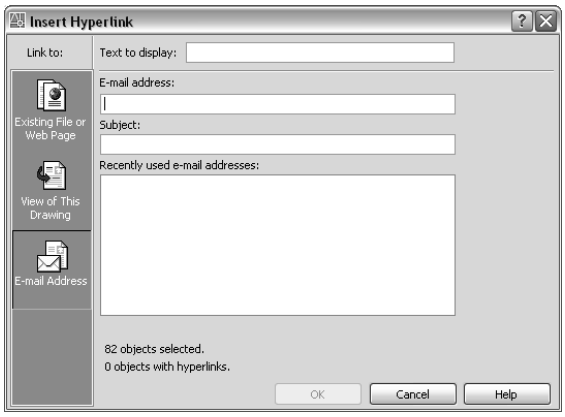
Specifies a named view in the current drawing to link to.




**Select a View of This** Displays an expandable tree view of the named views within the current drawing, from which you can select one to link to.

### E-mail Address Tab (Insert Hyperlink Dialog Box)

Specifies an email address to link to. When the hyperlink is executed, a new email is created using the default system email program.




**E-mail Address** Specifies an email address.

**Subject** Specifies a subject for the email.

**Recently Used E-mail Addresses** Lists recently used email addresses, from which you can select one to use for the hyperlink.

## Edit Hyperlink Dialog Box

### Quick Reference

Insert ➤ HyperlinkAt the Command prompt, enter hyperlink.
To edit a hyperlink, select an object that contains a hyperlink, right-click in the drawing area, and choose Hyperlink ➤ Edit Hyperlink.
**hyperlink**

Edits hyperlinks. The Edit Hyperlink dialog box is a modified version of the Insert Hyperlink dialog box (page 647), with one additional option.

**Remove Link** Deletes the hyperlink from the selected objects.

## Select Place in Document Dialog Box

### Quick Reference

Insert ➤ HyperlinkAt the Command prompt, enter hyperlink.

To edit a hyperlink, select an object that contains a hyperlink, right-click in the drawing area, and choose Hyperlink ➤ Edit Hyperlink.
**hyperlink**

Navigates to a view or layout within a drawing. The named location that you select is the initial view that is restored when the hyperlink is executed.

**Select an Existing Place in the Document** Provides an expandable tree view of the named locations within the linked drawing, from which you can select a named location to link to.

# -HYPERLINK

## Quick Reference

If you enter **-hyperlink** at the command prompt, the following HYPERLINK command prompts are displayed. The command prompt version of HYPERLINK inserts hyperlinks to areas or to selected objects and also removes hyperlinks.

Enter an option [Remove (page 651)/Insert (page 651)] <Insert>: *Enter an option or press*ENTER

### Remove

Removes the hyperlink from the selected objects. If a selection set does not exist, you are prompted to create one.

Select objects: *Select graphical objects in the drawing*

If the selection set contains more than one hyperlink, the hyperlinks that were found are listed at the command prompt.

Enter number, hyperlink, or * for all: *Enter the number or name of the hyperlink, or enter***to remove all hyperlinks in the selection set*

The number of removed hyperlinks is displayed at the command prompt.

### Insert

Attaches a hyperlink to an object or an area.

Enter hyperlink insert option [Area/Object] <Object>: *Select a method for attaching a hyperlink*
**Area** Defines a bounding rectangle and places it on a layer called URLLAYER. If this layer doesn't exist in the current drawing, it is created.

**NOTE** To turn off the display of hyperlink bounding rectangles in the drawing area or in your drawing plots, you must freeze the URLLAYER layer. See "Use Layers to Manage Complexity" in the *User's Guide*.

First corner: *Specify the start point of the rectangle*

Other corner: *Specify the end point of the rectangle*

Enter the path to the file you want associated with the hyperlink.

Enter hyperlink <*current drawing*>: *Enter the full path to a local file or the complete URL to an Internet file, or press* ENTER *to create a hyperlink to a named view in the current drawing*

Enter a named location for the hyperlink, such as a view in a drawing or a bookmark in a word processing file.

Enter named location <none>: *Enter the named location, or press* ENTER *to create the hyperlink without one*

Enter description <none>: *Enter a description, or press* ENTER *to create the hyperlink without one*

**Object** Specifies graphical objects to attach the hyperlink to.

Select objects: *Select graphical objects in the drawing*

Enter the path to the file you want associated with the hyperlink.

Enter hyperlink <current drawing>: *Enter the full path to a local file or the complete URL to an Internet file, or press* ENTER *to create a hyperlink to a named view in the current drawing*

Enter a named location for the hyperlink, such as a view in a drawing or a bookmark in a word processing file.

Enter named location <none>: *Enter the named location, or press* ENTER *to create the hyperlink without one*

Enter description <none>: *Enter a descriptive name for the hyperlink, or press* ENTER *to create the hyperlink without one*

# HYPERLINKOPTIONS

## Quick Reference

Controls the display of the hyperlink cursor, tooltips, and shortcut menu

**hyperlinkoptions**
Display hyperlink cursor, tooltip, and shortcut menu? [Yes/No] <Yes>:

**NOTE** Access to hyperlinks is not available if you choose No.

# I Commands

# 10

**In this chapter**

- ID
- IMAGE
- IMAGEADJUST
- IMAGEATTACH
- IMAGECLIP
- IMAGEFRAME
- IMAGEQUALITY
- IMPORT
- IMPRINT
- INSERT
- INSERTOBJ
- INTERFERE
- INTERSECT
- ISOPLANE

# ID

## Quick Reference

Displays the coordinate of a location

Tools ➤ Inquiry ➤ ID PointAt the Command prompt, enter id.
**id** (or '**id** for transparent use)
Point: *Use the pointing device to specify a point*

The UCS coordinate of location is displayed at the command prompt.

ID lists the *X*, *Y*, and *Z* values of the specified point and stores the coordinate of the specified point as the last point. You can reference the last point by entering **@** at the next prompt that requests a point.

If you snap to an object in 3D space, the *Z* coordinate value is the same as that of the selected feature of the object.

# IMAGE

## Quick Reference

Displays the External References palette

Reference 
Insert ➤ External ReferencesAt the Command prompt, enter externalreferences.
Select an image, right-click in the drawing area, and choose Image ➤ External References.
**image**

The External References palette (page 561) is displayed.

If you enter **-image** at the command prompt, options are displayed at the command prompt (page 657).

# -IMAGE

## Quick Reference

If you enter **-image** at the command prompt, the following IMAGE command prompts are displayed.

Enter image option [? (page 657)/Detach (page 657)/Path (page 657)/Reload (page 657)/Unload (page 658)/Attach (page 658)] <Attach>: *Enter an option or press* ENTER

### ?—List Images

Lists the images by name in alphabetical order, the number of times each is attached to the drawing, and the path where the image is stored. Images are listed in alphabetical order, regardless of the setting of the *MAXSORT* system variable.

Images to list <*>: *Enter * to list all images, or enter an image name*

### Detach

Detaches the named image from the drawing, marks it for deletion, and erases all occurrences of the image.

Enter list of images to detach: *Enter an image name, or enter *to detach all images*

### Path

Updates the path name (including file name) associated with a particular image. This option is useful if you change the location of an image file, rename the file, or replace an old image file with a new file; for instance, you can update *image01.pcx* and save it as *image02.pcx*.

Enter list of images for path modification: *Enter an image name, or enter * to list all images*

If you enter an asterisk (*), the following prompt is displayed:

Old path: *Lists the current path name for each image*
Enter New path: *Enter the new path name for the specified image*

### Reload

Reloads the selected images, making that information available for display and plotting.

Enter list of images to reload: *Enter an image name, or enter * to reload all images*
Reloading...
Reload image <*image name*>: <*hard-coded path name*>
<*image name*> loaded and relinked.

### Unload

Removes image data from working memory so that the images are not displayed, thus improving performance. All information associated with the image remains stored with the drawing. The image frame of each attached image remains visible.

Enter list of images to unload: *Enter a loaded image name, or enter * to unload all images*

### Attach

Attaches a new image or a copy of an attached image to the current drawing. The Select Image File dialog box (a standard file selection dialog box (page 931)) is displayed.

The file name without the extension is assigned as the image name. Image names can include up to 255 characters and contain letters, digits, spaces, and any special characters not used by Microsoft Windows or this program. If the file name is not a valid name for a nongraphical object, the Substitute Image Name dialog box is displayed. A valid image name is generated from the file name and an underscore and number are appended to the name. The number changes as necessary to avoid duplicate image names.

Image *file name_1* created from file *file name.bmp*

If a definition with the same name and path exists in the drawing, the following prompts are displayed and the image is inserted as a copy:

Image file name has already been loaded.
Use IMAGE Reload to update its definition.
Specify insertion point <0,0>: *Specify an insertion point*
Base image size: Width: *current width*, Height: *current height*, *current unit*
Specify scale factor: *Enter a value or press* ENTER
Specify rotation angle <0>: *Enter a value or press* ENTER

If the *FILEDIA* system variable is set to 0, the following prompt is displayed instead of the dialog box:

Enter image file name to attach <*last*>: *Enter an image name*

The last image name attached to the drawing during the current session is the default. To avoid errors when entering an image name, it is recommended that you specify both the image name and the file name as follows:

*imagename=path name\long file name.bmp*

or

*imagename="path name\long file name.bmp"*

If you enter a valid image name without a file extension, the program searches for the file in this order: first, an existing image definition in the drawing, and second, an image file in the folders in order of the search path. The program searches for all the image files with the specified name, regardless of extension, and uses the first name found. If no image name or image file is found, the message "Image Not Found" is displayed and the prompt is repeated.

To specify a long file name that does not conform to this program's naming rules, enter the name as follows:

"*imagename=filename*"

You can use a dialog box to search for image files but still enter the *imagename=filename* convention at the command prompt. Enter a tilde (~) at the Enter Image File Name to Attach prompt. If you press ESC after the dialog box opens, the Enter Image Name prompt is displayed.

# IMAGEADJUST

## Quick Reference

Controls the image display of the brightness, contrast, and fade values of images

Reference
Modify ➤ Object ➤ Image ➤ AdjustAt the Command prompt, enter imageadjust.
Select an image to adjust, right-click in the drawing area, and choose Image ➤ Adjust.
**imageadjust**
Select image(s): *Select one or more images*

The Image Adjust dialog box (page 660) is displayed.

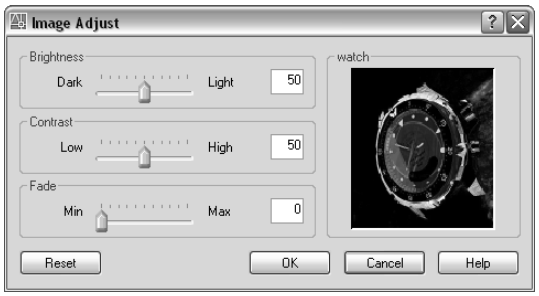
If you enter **-imageadjust** at the command prompt, options are displayed at the command prompt (page 661).

# Image Adjust Dialog Box

## Quick Reference

Reference
Modify ➤ Object ➤ Image ➤ AdjustAt the Command prompt, enter imageadjust.
Select an image to adjust, right-click in the drawing area, and choose Image ➤ Adjust.
**imageadjust**

Controls how the image is displayed by adjusting the brightness, contrast, and fade settings of the selected image. Adjusting these values changes the display of the image but does not change the image file itself.




**Brightness** Controls the brightness, and indirectly the contrast, of the image. Values range from 0 through 100. The greater the value, the brighter the image and the more pixels that become white when you increase contrast. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

**Contrast** Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

**Fade** Controls the fading effect of the image. Values range from 0 through 100. The greater the value, the more the image blends with the current background color. A value of 100 blends the image completely into the

background. Changing the screen background color causes the image to fade to the new color. In plotting, the background color for fade is white. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

**Image Preview** Displays a preview of the selected image. The preview image updates dynamically to reflect changes to the brightness, contrast, and fade settings.

**Reset** Resets values for brightness, contrast, and fade to default settings (50, 50, and 0, respectively).

# -IMAGEADJUST

## Quick Reference

If you enter **-imageadjust** at the command prompt, the following IMAGEADJUST command prompts are displayed. With the command prompt version of IMAGEADJUST, you can select more than one image to adjust.

Select image (s): *Select one or more images*
Enter image option [Contrast (page 661)/Fade (page 661)/Brightness (page 661)] <Brightness>: *Enter* **c** *or* **f**, *or press* ENTER *to adjust brightness*

If you selected a single image, the default values for Brightness, Contrast, and Fade are the current property settings of the image selected. If you selected multiple images, the default values for Brightness, Contrast, and Fade are 50, 50, and 0, respectively.

**Contrast** Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color.

Enter contrast value (0-100) <50>: *Enter a value*

**Fade** Controls the fading effect of the image. Values range from 0 through 100. The greater the value, the more the image blends with the current background color. A value of 100 blends the image completely into the background. Changing the screen background color causes the image to fade to the new color. In plotting, the background color for fade is white.

Enter fade value (0-100) <0>: *Enter a value*

**Brightness** Controls the brightness, and indirectly the contrast, of the image. Values range from 0 through 100. The greater the value, the brighter the image and the more pixels that become white when you increase contrast.

Enter brightness value (0-100) <50>: *Enter a value*

# IMAGEATTACH

## Quick Reference

Attaches a new image to the current drawing

Reference 
Insert ➤ Raster Image ReferenceDoes not exist in the menus.
**imageattach**

The Select Image File dialog box (a standard file selection dialog box (page 931)) is displayed. Once you select an image file, the Image dialog box (page 662) is displayed.

---

**NOTE** When Texture Compression is turned on, the amount of video memory required to open a drawing that contains attached images is decreased. By using the effect it can reduce the amount of video memory necessary to display the drawing, but the downside to this effect is it may increase the time it takes to load the images the first time that they are accessed and there is a reduction in the quality of the images when they are displayed in the viewport or plotted. To identify if Texture Compression is enabled, enter **3dconfig**, and click Manual Tune. Look at the Hardware Effects List in the Manual Performance Tuning dialog box.

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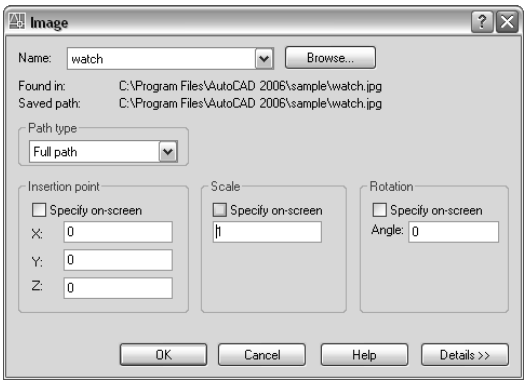
# Image Dialog Box

## Quick Reference

Reference
Insert ➤ Raster Image ReferenceDoes not exist in the menus.
**imageattach**

Locates, inserts, names, and defines the parameters and details of attached images.




### Name

Identifies the image you have selected to attach, either from the Select Image File dialog box (an unattached image) or from the list of previously attached images. To add another instance of an image file that is already attached, select the image name from the list and click OK.

**Browse** Opens the Select Image File dialog box (a standard file selection dialog box (page 931)). If Show Preview is selected, a preview of the selected file is displayed.

**Found In** Displays the path where the image file was located.

**Saved Path** Displays the path specified when the image file was attached to the current drawing.

### Path Type

Specifies one of three types of folder path information to save with an attached image: an absolute path, a relative path, and no path. For a complete description of each option, see “Set Paths to Referenced Drawings” in the *User's Guide*.

**Full Path** Specifies the absolute path to the image file.

**Relative Path** Specifies a relative path to the image file.

**No Path** Specifies only the image file name. The image file should be located in the folder with the current drawing file.

### Insertion Point

Specifies the insertion point for the selected image. Specify On-Screen is the default. The default insertion point is 0,0.

**Specify On-Screen** Directs input to the command prompt or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.

**X** Sets the *X* coordinate value.

**Y** Sets the *Y* coordinate value.

**Z** Sets the *Z* coordinate value.

### Scale

Specifies the scale factor of the selected image. Specify On-Screen directs input to the command prompt or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.

If *INSUNITS* is set to "unitless" or if the image does not contain resolution information, the scale factor becomes the image width in AutoCAD units. If INSUNITS has a value such as millimeters, centimeters, inches, or feet, and the image has resolution information, the scale factor is applied after the true width of the image in AutoCAD units is determined.

### Rotation

Specifies the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value at the command prompt. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

### Details

Displays information about the selected image.

**Resolution** Displays information about the vertical and horizontal resolution of the selected image.

**Current AutoCAD Unit** Displays information about the default units of the selected image. The image is displayed at the default width and height in pixels.

**Image Size in Pixels** Displays information about the width and height in pixels of the selected image.

**Image Size in Units** Displays information about the default size in units of the selected image. The default value for unitless images is unitless. The image size is automatically converted to AutoCAD units and is displayed at the default width and height.

# IMAGECLIP

## Quick Reference

Uses clipping boundaries to define a subregion of an image object

Reference 
Modify ➤ Clip ➤ ImageAt the Command prompt, enter imageclip.
Select an image to clip, right-click in the drawing area, and choose Image ➤ Clip.
**imageclip**
Select image to clip: *Select an edge of an image*
Enter image clipping option [ON (page 665)/OFF (page 665)/Delete (page 665)/New boundary (page 666)] <New>: *Enter an option or press* ENTER

The boundary you specify must be in a plane parallel to the image object.

### On

Turns on clipping and displays the image clipped to the previously defined boundary.

### Off

Turns off clipping and displays the entire image and frame.

If you reclip the image while clipping is turned off, clipping is automatically turned back on. You are prompted to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.

### Delete

Removes a predefined clipping boundary and redisplays the full original image.

### New Boundary

Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the image boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the point is interpreted as the first corner of a rectangle.

Enter clipping type [Polygonal/Rectangular] <Rectangular>: *Enter* **p** *or press* ENTER

**Polygonal** Uses specified points to define a polygonal boundary.

Specify first point: *Specify a point*

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Undo]: *Specify a point or enter* **u**

Specify next point or [Close/Undo]: *Specify a point, or enter* **c** *or* **u**

You must specify at least three points to define a polygon.

If the image already has a clipping boundary defined, the following prompt is displayed:

Delete old boundary? [No/Yes] <Yes>: *Enter* **n** *or press* ENTER

If you choose Yes, the entire image is redrawn and the command continues; if you choose No, the command ends.

**Rectangular** Specifies a rectangular boundary by its opposite corners. The rectangle is always drawn parallel to the edges of the image.

Specify first corner point: *Specify a point*

Specify opposite corner point: *Specify a point*

# IMAGEFRAME

## Quick Reference

Controls whether image frames are displayed and plotted

Reference 

Modify ➤ Object ➤ Image FrameDoes not exist in the menus.

**imageframe**

Enter image frame setting [0, 1, 2] <*current*>: *Enter an option or press* ENTER

**NOTE** Normally, when image frames are not displayed, you cannot select images. However, the IMAGECLIP command temporarily turns image frames on.

**0 Setting** Image frames are not displayed and not plotted.

**1 Setting** Image frames are both displayed and plotted. This setting is the default.

**2 Setting** Image frames are displayed but not plotted.

# IMAGEQUALITY

## Quick Reference

Controls the display quality of images

Reference

Modify ➤ Object ➤ Image ➤ QualityAt the Command prompt, enter imagequality.

**imagequality**

Enter image quality setting [High/Draft] <*current*>: *Enter an option or press* ENTER

The quality setting affects display performance; high-quality images take longer to display. Changing the setting updates the display immediately without causing a regeneration of the drawing.

**NOTE** Images are always plotted using a high-quality setting.

**High** Produces a high-quality display of the image.

**Draft** Produces a lower-quality display of the image.

# IMPORT

## Quick Reference

Imports files in various formats

Insert 
**import**

The Import File dialog box (a standard file selection dialog box (page 931)) is displayed.

In Files of Type, select the file format to import. In File Name, select the file name to import. The file is imported into the drawing. The following formats are available for import:

- WMF—Windows metafile (see *WMFIN*)
- SAT—ACIS solid object file (see *ACISIN*)
- 3DS—3D Studio file (see *3DSIN*)
- V8 DGN—MicroStation DGN file (see *DGNIMPORT*)

For the WMF file type, choosing Options from the Tools menu in this dialog box displays the WMF In Options dialog box (page 1562). You can display this dialog box directly by using *WMFOPTS*.

If *FILEDIA* = 0, the following prompt is displayed:

Enter import file name: *Enter path and file name*

# IMPRINT

## Quick Reference

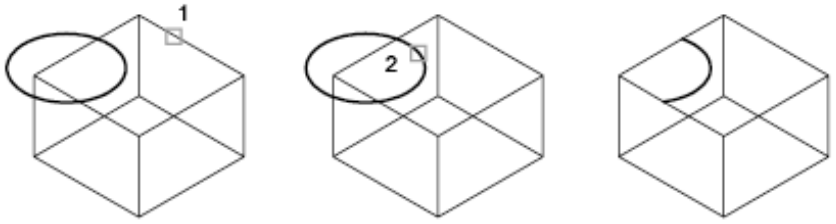
Imprints an edge on a 3D solid

Modify ➤ Solid Editing ➤ Imprint EdgesAt the Command prompt, enter imprint.
**imprint**
3D Make panel, Imprint

Imprints an object on the selected solid. The object to be imprinted must intersect one or more faces on the selected solid in order for imprinting to be successful. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids.

Select a 3D solid: *Select an object (1)*

Select an object to imprint: *Select an object (2)*
Delete the source object <N>: *Enter* **y** *or press* ENTER
Select an object to imprint: *Select an object or press* ENTER




# INSERT

## Quick Reference

Places a drawing or named block into the current drawing

Insert 
Insert ➤ BlockAt the Command prompt, enter insert.
**insert**

The Insert dialog box (page 669) is displayed.

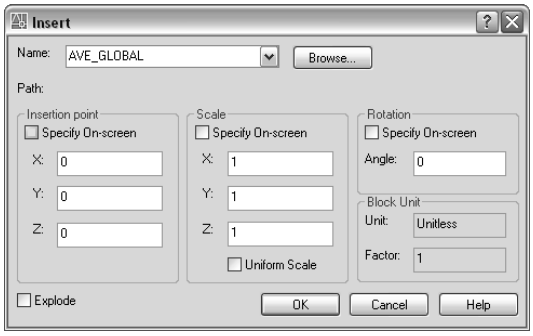
If you enter **-insert** at the command prompt, options are displayed at the command prompt (page 672).

# Insert Dialog Box

## Quick Reference

Insert
Insert ➤ BlockAt the Command prompt, enter insert.
**insert**

Specifies the name and position of the block or drawing to insert.




The last block you insert during the current editing session becomes the default block for subsequent uses of INSERT. The position of the inserted block depends on the orientation of the UCS.

### Name

Specifies the name of a block to insert, or the name of a file to insert as a block.

**Browse** Opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or drawing file to insert.

**Path** Specifies the path to the block.

**Preview** Displays a preview of the specified block to insert. A lightning bolt icon in the lower-right corner of the preview indicates that the block is dynamic. A  icon indicates that the block is .

### Insertion Point

Specifies the insertion point for the block.

**Specify On-Screen** Specifies the insertion point of the block using the pointing device.

**X** Sets the *X* coordinate value.

**Y** Sets the *Y* coordinate value.

**Z** Sets the *Z* coordinate value.

### Scale

Specifies the scale for the inserted block. Specifying negative values for the *X*, *Y*, and *Z* scale factors inserts a mirror image of a block.

**Specify On-Screen** Specifies the scale of the block using the pointing device.

**X** Sets the *X* scale factor.

**Y** Sets the *Y* scale factor.

**Z** Sets the *Z* scale factor.

**Uniform Scale** Specifies a single scale value for *X*, *Y*, and *Z* coordinates. A value specified for *X* is also reflected in the *Y* and *Z* values.

### Rotation

Specifies the rotation angle for the inserted block in the current UCS.

**Specify On-Screen** Specifies the rotation of the block using the pointing device.

**Angle** Sets a rotation angle for the inserted block.

### Block Unit

Displays information about the block units.

**Unit** Specifies the *INSUNITS* value for the inserted block.

**Factor** Displays the unit scale factor, which is calculated based on the *INSUNITS* value of the block and the drawing units.

### Explode

Explodes the block and inserts the individual parts of the block. When Explode is selected, you can specify a uniform scale factor only.

Component objects of a block drawn on layer 0 remain on that layer. Objects having color BYBLOCK are white. Objects with linetype BYBLOCK have the CONTINUOUS linetype.

# -INSERT

## Quick Reference

If you enter **-insert** at the command prompt, the following INSERT command prompts are displayed.

Enter block name (page 672) or [? (page 673)] <*last*>: *Enter a name, enter* **?**, *enter ~, or press* ENTER
Units: <INSUNITS (page 1727) *specified for inserted block*> Conversion: <*conversion scale*>
Specify insertion point (page 673) or [Basepoint (page 674)/Scale (page 674)/X (page 674)/Y (page 674)/Z (page 675)/Rotate (page 675)]: *Specify a point or enter an option*

### Block Name

If you have inserted a block in the current drawing during the current editing session, the name of the last block inserted appears as the current block in the prompt.

Grouped objects in an inserted drawing are inserted as unnamed groups. You can list unnamed groups by selecting Unnamed Groups in the Object Grouping dialog box (page 609).

Entering a tilde (~) displays the Select Drawing File dialog box (a standard file selection dialog box (page 931)).

You can control block insertion behavior in response to the Enter Block Name prompt by following the listed examples.

- *Inserting Exploded Blocks:* Preceding the name of the block with an asterisk (*) explodes the block and inserts the individual parts of it. The block definition is not added to the drawing.
- *Updating a Block Path:* If you enter a block name without a path name, INSERT searches the current drawing data for an existing block definition by that name. If no such block definition exists in the current drawing, INSERT searches the library path. If a file is found, the file name is used for the block name upon insertion of the block definition. The same block definition is used for subsequent insertions of that block. You can replace an existing block definition with an external file by entering the following at the Enter Block Name prompt:
  **block name=file name**

- *Updating a Block Definition:* If you make changes to a block file that is inserted in your drawing and you want to change the existing block definition without creating a new block insertion, enter the following at the Specify Insertion Point prompt (following the Enter Block Name prompt):
  **block name=**

  If you enter = after the block name, the following prompt is displayed:

  Block "current" already exists. Redefine it? [Yes/No] <No>: *Enter* **y**, *enter* **n**, *or press* ENTER

  If you choose to redefine the block, the existing block definition is replaced with the new block definition. The drawing is regenerated, and the new definition is applied to all existing insertions of the block definition. Press ESC when prompted for the insertion point if you do not want to insert a new block into the drawing.

### ?—List Block Names

Lists the blocks currently defined in the drawing.

### Insertion Point

Specifies a location for the block or drawing.

Specify scale factor or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER
**Scale Factor** Sets the scale factor.

Specify rotation angle <0>:

All *X* and *Y* dimensions of the block or drawing are multiplied by the *X* and *Y* scale factors. The block or drawing is rotated by the specified angle, using the insertion point as the center of rotation.

**Corner** Defines the *X* and *Y* scale factors at the same time, using the insertion point and another point as the corners of a box. The *X* and *Y* dimensions of the box become the *X* and *Y* scale factors. The insertion point is the first corner.

Specify opposite corner: *Specify a point*

Specify rotation angle <0>:

**XYZ** Sets *X*, *Y*, and *Z* scale factors.

Specify X scale factor or [Corner] <1>: *Specify a nonzero value, enter* **c**, *or press* ENTER

- *X Scale Factor:* Defines *X*, *Y*, and *Z* scale factors for the block or drawing. Specify Y scale factor or <use X scale factor>: *Enter a scale factor or press* ENTER

Enter Z scale factor <use X scale factor>: *Enter a scale factor or press* ENTER

Specify rotation angle <0>: *Specify an angle or press* ENTER

- *Corner:* Defines the *X* and *Y* scales at the same time, using the insertion point and another point as the corners of a box, and then defines the *Z* scale.
  Specify opposite corner: *Specify a point*

  Enter Z scale factor <use X scale factor>: *Enter a scale factor or press* ENTER

  Specify rotation angle <0>: *Specify an angle or press* ENTER

### Basepoint

Temporarily drops the block in the drawing where it is currently positioned and allows you to specify a new base point for the block reference as it is dragged into position. This does not affect the actual base point defined for the block reference.

Specify base point: *Specify a base point for the block reference*

### Scale

Sets the scale factor for the *X*, *Y*, and *Z* axes. The scale for the *Z* axis is the absolute value of the specified scale factor.

Specify scale factor for XYZ axes: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

### X

Sets the *X* scale factor.

Specify X scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

### Y

Sets the *Y* scale factor.

Specify Y scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

#### Z

Sets the *Z* scale factor.

Specify Z scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

#### Rotate

Sets the angle of insertion for the block.

Specify rotation angle <0>:
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options under Rotation match the descriptions of the corresponding options under Insertion Point.

# INSERTOBJ

### Quick Reference

Inserts a linked or embedded object

Insert 
Insert ➤ OLE ObjectAt the Command prompt, enter insertobj.
**insertobj**

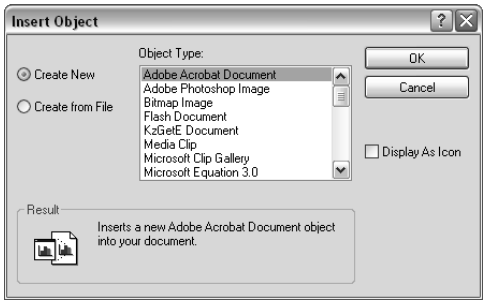
The Insert Object dialog box (page 675) is displayed.

## Insert Object Dialog Box

### Quick Reference

Insert
Insert ➤ OLE ObjectAt the Command prompt, enter insertobj.
**insertobj**

Inserts a linked or embedded object. Options vary, depending on whether you select Create New or Create from File.




### Create New

Opens the application that's highlighted in the Object Type list so that you can create a new object to insert.

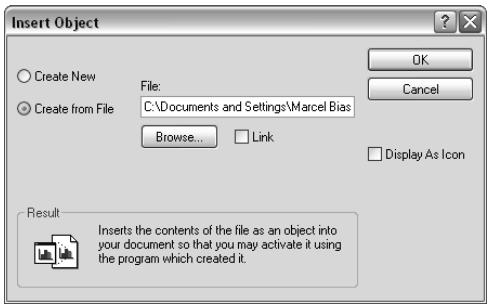
**Object Type** Lists available applications that support linking and embedding. To create an object to embed, double-click an application to open it.

On the application's File menu, the Save option is replaced with a new Update option. Choosing Update inserts the object into the drawing or updates it.

**Display as Icon** Displays the source application's icon in the drawing. Double-clicking the icon displays the embedded information.

### Create from File

Specifies a file to link or embed.




**File** Specifies the path and name of the file to be embedded or linked.

**Browse** Displays the Browse dialog box (a standard file selection dialog box), in which you select a file to link or embed.

**Link** Creates a link to the selected file rather than embedding it.

**Display as Icon** Displays the source application's icon in the drawing. Double-clicking the icon displays the linked or embedded information.

# INTERFERE

## Quick Reference

Highlights 3D solids that overlap

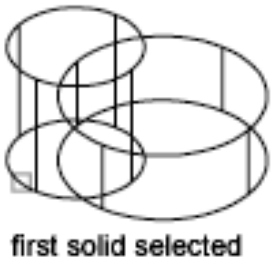
Modify ➤ 3D Operations ➤ Interference CheckingDoes not exist in the menus.
**interfere**
3D Make panel (click icon to expand), Interference Checking
Select first set of objects or [Nested selection (page 678)/Settings (page 678)]: *Use an object selection method or enter an option*

INTERFERE highlights 3D solids that overlap by creating a temporary composite 3D solid from the common volume of two or more solids. If you define a single selection set, INTERFERE checks all the solids in the set against one another. If you define two selection sets, INTERFERE checks the solids in the first selection set against those in the second selection set. If you include the same 3D solid in both selection sets, INTERFERE considers the 3D solid part of the first selection set and ignores it in the second selection set.

Select second set of objects or [Nested selection (page 678)/checK first set (page 678)] <checK>: *Use an object selection method, press* ENTER *to check for interferences, or enter* **n**

Pressing ENTER starts the interference testing of pairs of 3D solids and displays the Interference Checking dialog box (page 679).

If you enter **-interfere** at the command prompt, options are displayed at the command prompt (page 681).




#### Nested Selection

Allows you to select individual solid objects that are nested in blocks and xrefs.

Select nested objects or [eXit] <X>: *Select nested objects or press* ENTER *to return to normal object selection*

#### Settings

The Interference Settings dialog box (page 678) is displayed.

#### Check

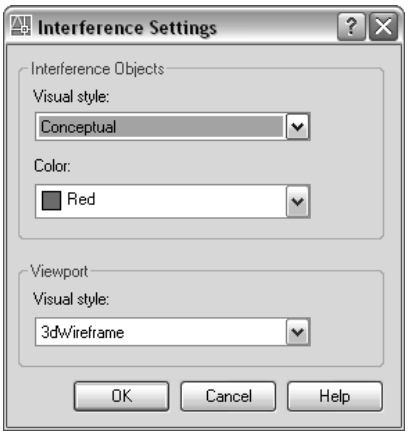
The Interference Checking dialog box (page 679) is displayed.

## Interference Settings Dialog Box

### Quick Reference

Modify ➤ 3D Operations ➤ InterferenceDoes not exist in the menus.
**interfere**

Controls the display of interference objects.




### Interference Objects

Specifies the visual style and color for interference objects.

**Visual Style** Specifies the visual style for interference objects. (*INTERFEREOBJVS*)

**Color** Specifies the color for interference objects. (*INTERFERECOLOR*)

**Highlight Interfering Pair** Highlights the interfering pair of solids.

**Highlight Interference** Highlights the interference objects created from the interfering pair.

### Viewport Display

Specifies the viewport display while checking for interferences.

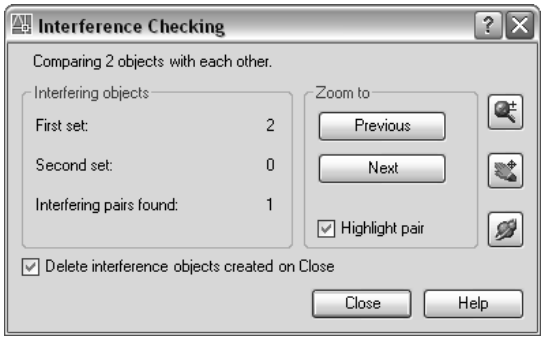
**Visual Style** Specifies the visual style for the viewport while checking for interferences. (*INTERFEREVPVS*)

## Interference Checking Dialog Box

### Quick Reference

Modify ➤ 3D Operations ➤ InterferenceDoes not exist in the menus.
**interfere**

Allows you to cycle through and zoom to interference objects. You can also specify whether or not to delete the interference objects when you close the dialog box.




### Interfering Objects

Displays the number of interferences found between each set during the *INTERFERE* command.

**First Set** Displays the number of objects selected in the first set.

**Second Set** Displays the number of objects selected in the second set.

**Interfering Pairs Found** Displays the number of interferences found among the selected objects.

### Highlight

Highlights interference objects while using Previous and Next to cycle through the objects.

**Previous** Highlights the previous interference object.

**Next** Highlights the next interference object.

**Zoom to Pair** Zooms to interference objects while using Previous and Next.

### Zoom

Closes the dialog box and starts the *ZOOM* command.

### Pan

Closes the dialog box and starts the *PAN* command.

#### 3D Orbit

Closes the dialog box and starts the *3DORBIT* command.

#### Delete Interference Objects Created on Close

Deletes the interference objects when the dialog box is closed.

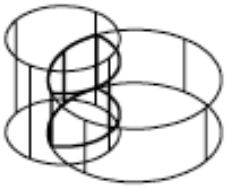
#### Close

Closes the dialog box and deletes the interference objects if that option is selected.

# -INTERFERE

## Quick Reference

If you enter **-interfere** at the command prompt, the following INTERFERE command prompt are displayed.



interference solid created

Check for interferences between 2 sets of objects or within 1 set of objects...
Select first set of objects or [Nested selection (page 682)]: *Use an object selection method or enter an option*
Select second set of objects or [Nested selection (page 682)/checK current] <checK>: *Use an object selection method, enter* **n**, *or press* ENTER *to check for interferences*

INTERFERE highlights all interfering 3D solids and displays the number of objects selected and the number of interfering pairs.

If you define a single selection set, INTERFERE checks all the solids in the set against one another. If you define two selection sets, INTERFERE checks the solids in the first selection set against those in the second selection set. If you

include the same 3D solids in both selection sets, INTERFERE considers the 3D solid part of the first selection set and ignores it in the second selection set.

Create interference objects? [Yes/No] <N>: *Enter* **y** *or* **n***, or press* ENTER to select the default option

Entering **y** creates and highlights new 3D solids on the current layer that are the intersections of the interfering pairs of 3D solids.

If there are more than two interfering 3D solids, it may not be clear which pairs are interfering if all the interfering 3D objects are highlighted at once.

Zoom to pairs of interfering objects? [Yes/No] <N>: *Enter* **y** *or* **n***, or press* ENTER to select the default option

If there is more than one interfering pair, the following prompt is displayed:

Enter an option [Next pair/eXit] <Next>: *Enter* **x** *or* **n***, or press* ENTER

Entering **n** or pressing ENTER cycles through the interfering pairs of 3D solids. Entering **x** ends the command.

#### Nested Selection

Allows you to select individual solid objects that are nested in blocks and xrefs.

Select nested objects or [eXit] <X>: *Select nested objects or press* ENTER *to return to normal object selection*

# INTERSECT

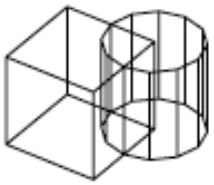
## Quick Reference

Creates composite solids or regions from the intersection of two or more solids or regions and removes the areas outside of the intersection

Modeling 
Modify ➤ Solid Editing ➤ IntersectAt the Command prompt, enter intersect.
**intersect**
3D Make panel, Intersect
Select objects: *Use an object selection method*

You can select only regions and solids for use with INTERSECT.

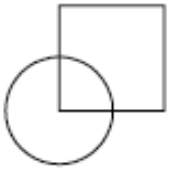
INTERSECT calculates the overlapping area of two or more existing regions and the common volume of two or more existing solids.



solids before INTERSECT

solid after INTERSECT

The selection set can contain regions and solids that lie in any number of arbitrary planes. INTERSECT divides the selection set into subsets and tests for intersections within each subset. The first subset contains all the solids in the selection set. The second subset contains the first selected region and all subsequent coplanar regions. The third subset contains the next region that is not coplanar with the first region and all subsequent coplanar regions, and so on until all regions belong to a subset.



regions before INTERSECT

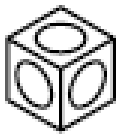
region after INTERSECT

# ISOPLANE

## Quick Reference

Specifies the current isometric plane

**isoplane** (or '**isoplane** for transparent use)
Enter isometric plane setting [Left (page 684)/Top (page 684)/Right (page 684)] <Top>: *Enter an option or press* ENTER



The isometric plane affects the cursor movement keys only when Snap mode is on and the snap style is Isometric. If the snap style is Isometric, Ortho mode uses the appropriate axis pair even if Snap mode is off. The current isometric plane also determines the orientation of isometric circles drawn by *ELLIPSE*. You can cycle through the isometric planes by pressing CTRL+E or F5.

**Left**

Selects the left-hand plane, defined by the 90-degree and 150-degree axis pair.

**Top**

Selects the top face of the cube, called the top plane, defined by the 30-degree and 150-degree axis pair.

**Right**

Selects the right-hand plane, defined by the 90-degree and 30-degree axis pair.

# J Commands

**In this chapter**

- JOGSECTION
- JOIN
- JPGOUT
- JUSTIFYTEXT

# JOGSECTION

## Quick Reference

Adds a jogged segment to a section object

jogsection
Select section object: *Select a section object*
Specify a point on the section line to add jog: *Pick a point on the section line*

A jog is created on the section line. The jogged segment is created at a 90 degree angle to the section line.

# JOIN

## Quick Reference

Joins objects to form a single, unbroken object

Modify 
Modify ➤ JoinAt the Command prompt, enter join.
**join**
Select source object: *Select a line, polyline, arc, elliptical arc, spline, or helix*

Depending on the source object selected, *one* of the following prompts is displayed:

### Line

Select lines to join to source: *Select one or more lines and press* ENTER

The line objects must be collinear (lying on the same infinite line), but can have gaps between them.

### Polyline

Select objects to join to source: *Select one or more objects and press* ENTER

The objects can be lines, polylines, or arcs. The objects cannot have gaps between them, and must lie on the same plane parallel to the UCS *XY* plane.

### Arc

Select arcs to join to source or [cLose]: *Select one or more arcs and press* ENTER, *or enter* **L**

The arc objects must lie on the same imaginary circle, but can have gaps between them. The Close option converts the source arc into a circle.

**NOTE** When joining two or more arcs, the arcs are joined counterclockwise beginning from the source object.

### Elliptical Arc

Select elliptical arcs to join to source or [cLose]: *Select one or more elliptical arcs and press* ENTER, *or enter* **L**

The elliptical arcs must lie on the same ellipse, but can have gaps between them. The Close option closes the source ellipstical arc into a complete ellipse.

**NOTE** When joining two or more elliptical arcs, the elliptical arcs are joined counterclockwise beginning from the source object.

### Spline

Select splines or helixes to join to source: *Select one or more splines or helixes and press* ENTER

The spline and helix objects must be contiguous (lying end-to-end). The resulting object is a single spline.

### Helix

Select splines or helixes to join to source: *Select one or more splines or helixes and press* ENTER

The helix objects must be contiguous (lying end-to-end). The resulting object is a single spline.

# JPGOUT

## Quick Reference

Saves selected objects to a file in JPEG file format

**jpgout**

The Create Raster File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the file name in the dialog box.

Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A JPEG file is created that contains the objects you select. Shade Plot options are preserved in the file when you use this command. Light glyphs that are displayed in the drawing appear in the new file, even if the Plot Glyph property of the lights is set to No.

**NOTE** When the *FILEDIA* system variable is set to 0 (Off), command prompts are displayed.

# JUSTIFYTEXT

## Quick Reference

Changes the justification point of selected text objects without changing their locations

Text 
Modify ➤ Object ➤ Text ➤ Justify.Does not exist in the menus.
**justifytext**
Select objects: *Use an object selection method and press* ENTER *when you finish*

You can choose single line text objects, multiline text objects, leader text objects, and attribute objects.

Enter a justification option
[Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]<Existing>:
*Specify a location to serve as the new justification point*

The justification point options shown above are described in the *TEXT* command. The justification point options for single line text are similar to those for multiline text except that the Align, Fit, and Left text options are equivalent to the bottom left (BL) multiline text attachment point.

# L Commands

# 12

**In this chapter**

- LAYWALK
- LEADER
- LENGTHEN
- LIGHT
- LIGHTLIST
- LIGHTLISTCLOSE
- LIMITS
- LINE
- LINETYPE
- LIST
- LIVESECTION
- LOAD
- LOFT
- LOGFILEOFF
- LOGFILEON
- LSEDIT
- LSLIB
- LSNEW
- LTSCALE
- LWEIGHT

# LAYCUR

## Quick Reference

Changes the layer of selected objects to the current layer

Layers II
Format ➤ Layer Tools ➤ Change to Current Layer
**laycur**
Select objects to be changed to the current layer: *Use an object selection methodand press* ENTER *when you are finished*
N object(s) changed to layer X (the current layer).

# LAYDEL

## Quick Reference

Deletes the layer of a selected object and all objects on the layer, and purges the layer from the drawing

Format ➤ Layer Tools ➤ Layer DeleteDoes not exist in the menus.
**laydel**
Select object on layer to delete or [Name]: *Select an object or enter* **n** *to display the Delete Layers dialog box*

If you enter **-laydel** at the command prompt, options are displayed at the command prompt (page 692).

### Select Object on Layer to Delete

Select an object on the layer you want to delete.

Select object on layer to delete or [Name/Undo]: *Select an object on a second layer or enter* **u**

### Name

Displays the Delete Layers dialog box (page 692).

## Delete Layers Dialog Box

### Quick Reference

Format ➤ Layer Tools ➤ Layer DeleteAt the Command prompt, enter laydel.
**laydel**

Displays layers available for deletion.

**Layers to Delete** Displays a list of layers that you can delete. Press SHIFT or CTRL to select more than one layer. Layers that cannot be deleted include the current layer, layer 0, locked layers, and xref-dependent layers.

## -LAYDEL

### Quick Reference

If you enter **-laydel** at the command prompt, the following LAYDEL command prompts are displayed.

Select object on layer to delete or [Name]: *Select an object or enter* **n.**
Select object on layer to delete or [Name/Undo]: *Select an object, enter* **n**, *or enter* **u**

#### Select Object on Layer to Delete

Select an object on the layer you want to delete.

Select object on layer to delete or [Undo]: *Select an object to delete or enter* **u**

#### Name

Select a layer to delete.

Enter layer name or [?]: *Enter a layer name or enter* **?**
Enter layer name(s) to list <*>: * *Enter a layer name or press ENTER to list all layers.*

# LAYER

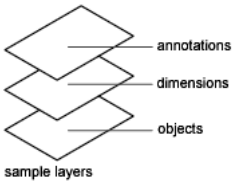
## Quick Reference

Manages layers and layer properties


Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

The Layer Properties Manager (page 693) is displayed.

If you enter **-layer** at the command prompt, options are displayed at the command prompt (page 710).

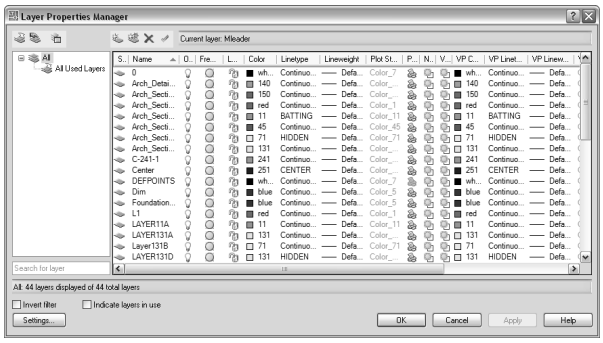



# Layer Properties Manager

## Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Displays a list of the layers in the drawing and their properties. You can add, delete, and rename layers, change their properties, set property overrides for layout viewports, or add descriptions. Layer filters control which layers are displayed in the list and can also be used to make changes to more than one layer at a time.




 **New Property Filter** Displays the Layer Filter Properties dialog box (page 702), where you can create a layer filter based on one or more properties of the layers.

 **New Group Filter** Creates a layer filter that contains layers that you select and add to the filter.

 **Layer States Manager** Displays the Layer States Manager (page 717), in which you can save the current property settings for layers in a named layer state and then restore those settings later.

 **New Layer** Creates a new layer. The list displays a layer named LAYER1. The name is selected so that you can enter a new layer name immediately. The new layer inherits the properties of the currently selected layer in the layer list (color, on or off state, and so on).

 **New Layer Frozen VP In All Viewports** Creates a new layer and freezes it in all existing layout viewports. This button is accessbile from the Model tab or layout tabs.


**Delete Layer** Marks selected layers for deletion. Layers are deleted when you click Apply or OK. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects (including objects in block definitions), the current layer, and xref-dependent layers.

Layers in a partially opened drawing are also considered referenced and cannot be deleted.

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**NOTE** Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

---

**Set Current**  Sets the selected layer as the current layer. Objects that you create are drawn on the current layer. (*CLAYER* system variable)

**Current Layer** Displays the name of the current layer.

**Search for Layer** Filters the layer list by name quickly as you enter characters. This filter is not saved when you close the Layer Properties Manager.

**Status Line** Shows the name of the current filter, the number of layers displayed in the list view, and the number of layers in the drawing.

**Invert Filter** Displays all layers that do not meet the criteria in the selected layer property filter.

**Indicate Layers in Use** Displays icons in the list view to indicate whether layers are in use. In a drawing with many layers, clear this option to improve performance. (*SHOWLAYERUSAGE* system variable)

**Settings** Displays the Layer Settings dialog box (page 708), in which you can set new layer notification settings, if layer filter changes are applied to the Layer toolbar, and change the background color for layer property overrides.

**Apply** Applies changes that have been made to layers and filters but does not close the dialog box.

The Layer Properties Manager has the following two panes:

- Tree view (page 696)
- List view (page 698)

### Tree View

Displays a hierarchical list of layers and filters in the drawing. The top node, All, displays all layers in the drawing. Filters are displayed in alphabetical order. The All Used Layers filter is read-only.

Expand a node to see nested filters. Double-click a property filter to open the Layer Filter Properties dialog box (page 702) and view the definition of the filter.

If there are xrefs attached to the drawing, an XREF node displays the names of all the xrefs in the drawing and the layers in each xref. Layer filters defined in xref files are not displayed.

If there are layers that contain property overrides, a Viewport Overrides node is automatically created and displays those layers and the properties that contain overrides. The Viewport Overrides filter displays only when the Layer Properties Manager is accessed from a layout tab.

If there are new layers that have been added to the drawing since the layer list was last evaluated (depending on how the LAYERNOTIFY system variable is set), an Unreconciled New Layers filter is automatically created and displays new layers that need to be reconciled.

### Tree View Shortcut Menu

Provides commands for items selected in the tree view.

**Visibility** Changes the visibility state of all the layers in the selected filter (or All or All Used Layers, if selected).

- *On:* Objects on the layer are displayed, plotted, and regenerated and hide other objects when you use *HIDE*.
- *Off:* Objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is not regenerated when you turn the layer on.
- *Thawed*: Objects on the layer are displayed and plotted and hide other objects when you use HIDE.
- *Frozen:* Objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is regenerated when you thaw the layer.

**Lock** Controls whether the objects on the layers in the selected filter can be modified.

- *Lock:* None of the objects on the layer can be modified. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.
- *Unlock:* Objects on the layer can be modified.

**Viewport** In the current layout viewport, controls the VP Freeze setting of the layers in the selected layer filter. This option is not available for model space viewports.

- *Freeze:* Sets VP Freeze for layers in the filter. In the current viewport, objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is regenerated when you thaw the layer.
- *Thaw:* Clears VP Freeze for layers in the filter. In the current viewport, objects on the layer are displayed and plotted and hide other objects when you use HIDE. This option does not thaw layers that are set to Off or Frozen in the drawing.

**Isolate Group** Turns off all layers not in the selected filter. The layers that are visible are the layers in the filter.

- *All Viewports:* In all viewports in a layout, sets VP Freeze for all layers that are not in the selected filter. In model space, turns off all layers that are not in the selected filter.
- *Active Viewport Only:* In the current layout viewport, sets VP Freeze for all layers that are not in the selected filter. In model space, turns off all layers that are not in the selected filter.

**New Properties Filter** Displays the Layer Filter Properties dialog box (page 702), where you can create a new layer filter based on layer names and settings; for example, on or off, color, or linetype.

**New Group Filter** Creates a new layer group filter named GROUP FILTER1 and adds it to the tree view. Enter a new name. Select the All filter or any other layer filter in the tree view to display layers in the list view, and then drag layers from the list view into the new layer group filter in the tree view.

You can use standard selection methods. Hold down CTRL to select more than one layer name. Hold down SHIFT and select the first and last layer names in a sequence to select all layers in the sequence.

**Convert to Group Filter** Converts the selected layer property filter to a layer group filter. Changing the properties of the layers in a layer group filter has no effect on the filter.

**Rename** Renames the selected filter. Enter a new name.

**Delete** Deletes the selected layer filter. You cannot delete the All, All Used Layers, or Xref filters. This option deletes the layer filter but not the layers in the filter.

**Properties** Displays the Layer Filter Properties dialog box (page 702), where you can modify the definition of the selected layer property filter. This option is available only when a layer property filter is selected.

**Select Layers** Temporarily closes the Layer Filter Properties dialog box so that you can select objects in the drawing. This option is available only when a layer group filter is selected.

- *Add:* Adds the layers of the selected objects to the layer group filter that is selected in the tree view.
- *Replace:* Replaces the layers of the selected layer group filter with the layers of the objects that are selected in the drawing.

### List View

Displays layers and layer filters and their properties and descriptions. If a layer filter is selected in the tree view, the list view displays only the layers in that layer filter. The All filter in the tree view displays all layers and layer filters in the drawing. When a layer property filter is selected and there are no layers that fit its definition, the list view is empty. To modify a property of a selected layer or of all layers in a selected filter, click the icon for that property. When a mixed icon or "Varies" is displayed for a layer filter, the property is not the same for all layers in the filter.

**Status** Indicates the type of item: layer filter, layer in use, empty layer, or current layer.

**Name** Displays the name of the layer or filter. Press F2 to enter a new name.

**On** Turns the selected layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

**Freeze** Freezes the selected layers in all viewports, including the Model tab. You can freeze layers to speed up *ZOOM*, *PAN*, and many other operations;

improve object selection performance; and reduce regeneration time for complex drawings.

Objects on frozen layers are not displayed, plotted, hidden, rendered, or regenerated.

Freeze the layers you want to be invisible for long periods. If you plan to switch visibility settings frequently, use the On/Off setting to avoid regenerating the drawing. You can freeze a layer in all viewports, in the current layout viewport, or in new layout viewports as they are created.

**Lock** Locks and unlocks the selected layers. Objects on a locked layer cannot be modified.

**Color** Changes the color associated with the selected layers. Clicking the color name displays the Select Color dialog box (page 251).

**Linetype** Changes the linetype associated with the selected layers. Clicking the linetype name displays the Select Linetype dialog box (page 706).

**Lineweight** Changes the lineweight associated with the selected layers. Clicking the lineweight name displays the Lineweight dialog box (page 707).

**Plot Style** Changes the plot style associated with the selected layers. If you are working with color-dependent plot styles (the *PSTYLEPOLICY* system variable is set to 1), you cannot change the plot style associated with a layer. Clicking the plot style displays the Select Plot Style dialog box (page 1078).

**Plot** Controls whether the selected layers are plotted. If you turn off plotting for a layer, the objects on that layer are still displayed. Layers that are off or frozen are not plotted, regardless of the Plot setting.

**VP Freeze** Freezes selected layers in the current layout viewport. You can freeze or thaw layers in the current viewport without affecting layer visibility in other viewports.

VP Freeze is an override to the Thaw setting in the drawing. That is, you can freeze a layer in the current viewport if it's thawed in the drawing, but you can't thaw a layer in the current viewport if it's frozen or off in the drawing. A layer is not visible when it is set to Off or Frozen in the drawing.

**New VP Freeze** Freezes selected layers in new layout viewports. For example, freezing the DIMENSIONS layer in all new viewports restricts the display of dimensions on that layer in any newly created layout viewports but does not affect the DIMENSIONS layer in existing viewports. If you later create a viewport that requires dimensions, you can override the default setting by changing the current viewport setting.

**VP Color (available only from a layout tab)** Sets an override for the color associated with the selected layer for the active layout viewport.

**VP Linetype (available only from a layout tab)** Sets an override for the linetype associated with the selected layer for the active layout viewport.

**VP Lineweight (available only from a layout tab)** Sets an override for the lineweight associated with the selected layer for the active layout viewport.

**VP Plot Style (available only from a layout tab)** Sets an override for the plot style associated with the selected layer for the active layout viewport. Override settings are not visible in the viewport or plotted when the visual style in the drawing is set to Conceptual or Realistic. If you are working with color-dependent plot styles (the *PSTYLEPOLICY* system variable is set to 1), you cannot set a plot style override.

**Description** (Optional) Describes the layer or the layer filter.

### List View Shortcut Menu

Provides options for modifying the list and modifying selected layers and layer filters.

### Column Label Shortcut Menu

**Column Names** Lists all columns by name. Columns that are checked indicate they are displayed. Hidden column names are not checked. VP Freeze, VP Color, VP Linetype, VP Lineweight, and VP Plot Style are available only when a layout viewport is active.

**Customize** Displays the Customize Layer Columns dialog box (page 709), where you can specify which columns are hidden or displayed.

**Maximize All Columns** Changes the width of all columns to maximize display of the column content. This option is available on the shortcut menu that is displayed when you right-click a column heading.

**Maximize Column** Changes the width of the column to maximize display of the column content. This option is available on the shortcut menu that is displayed when you right-click a column heading.

**Restore All Columns to Defaults** Restores all columns to their default display and width settings. This option is available on the shortcut menu when you right-click a column heading.

### Layer Shortcut Menu

**Show Filter Tree** Displays the tree view. Clear this option to hide the tree view.

**Show Filters in Layer List** Displays filters at the top of the list view. The filters are listed in alphabetical order. Clear the check mark to display only layers in the list view.

**Set Current** Sets the selected layer as the current layer. (*CLAYER* system variable)

**New Layer** Creates a new layer. The list displays a layer named LAYER1. You can edit this layer name immediately. The new layer inherits the properties of the currently selected layer in the layer list (color, on or off state, and so on).

**Rename Layer** Renames the layer.

**Delete Layer** Deletes selected layers from the drawing file definition. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects (including objects in block definitions), the current layer, and xref-dependent layers.

---

**NOTE** Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

---

**Change Description** Adds a description or changes the text in the description for the selected layer or filter. The description for a filter is added to the filter and to all layers in the filter.

**Remove from Group Filter** Removes the selected layers from the group layer filter that is selected in the tree view.

**Reconcile Layer** Removes new layers from the Unreconciled New Layers filter. Available only when one or more unreconciled layers is selected.

**Remove Viewport Overrides For** Removes a single override property or all property overrides on selected layers (or all layers) for the current viewport or all viewports. Different options display in a flyout menu depending on where the cursor is located when the shortcut menu is accessed. To remove a single property override, right-click the property override. This option is only available on the layout tab.

**New Layer Frozen in All Viewports** Creates a new layer and freezes it in all existing layout viewports and new viewports.

**VP Freeze Layer in All Viewports** Freezes the selected layer in all existing layout viewports and new viewports.

**VP Thaw Layer in All Viewports** Thaws the selected layer in all existing layout viewports and new viewports.

**Select All** Selects everything displayed in the list view.

**Clear All** Removes selection from all items in the list view except the most recently selected layer or filter.

**Select All but Current** Selects everything displayed in the list view except the current layer.

**Invert Selection** Selects everything displayed in the list view except the items that are currently selected.

**Invert Layer Filter** Displays all layers that do not meet the criteria in the selected layer property filter.

**Layer Filters** Displays a list of layer filters, including All and All Used Layers. Click a filter to apply it to the list view.

**Save Layer States** Displays the New Layer State to Save dialog box (page 720), in which you save the layer state and layer property settings of all layers in a drawing.

**Restore Layer State** Displays the Layer States Manager (page 717), in which you can select a named layer state to restore settings of all layers in the drawing. Restores only those layer state and property settings that were selected when the named layer state was saved.

# Layer Filter Properties Dialog Box

## Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Filters layers based on criteria you select. When a layer filter is selected in the tree view of the Layer Properties Manager (page 693), layers that match the filter criteria are displayed in the list view.

### Filter Name

Provides a space to enter a name for the layer properties filter.

### Show Example

Displays the examples of layer property filter definitions in "Layer Filter Examples".

### Filter Definition

Displays the properties of layers. You can use one or more properties to define the filter. For example, you can define a filter that displays all layers that are either red or blue and in use. To include more than one color, linetype, or lineweight, duplicate the filter on the next line and select a different setting.

**Status** Click the In Use icon or the Not In Use icon.

**Name** Use wild-card characters to filter layer names. For example, enter ***mech*** to include all layers with the letters *mech* in the name.

All the wild-card characters are listed in the table in "Filter and Sort the List of Layers" in the *User's Guide*.

**On** Click the On or the Off icon.

**Freeze** Click the Freeze or the Thaw icon.

**Lock** Click the Lock or the Unlock icon.

**Color** Click the [...] button to display the Select Color dialog box (page 251).

**Linetype** Click the [...] button to display the Select Linetype dialog box (page 706).

**Lineweight** Click the [...] button to display the Lineweight dialog box (page 707).

**Plot Style** Click the [...] button to display the Select Plot Style dialog box (page 1078).

**Plot** Click the Plot icon or the Do Not Plot icon.

**VP Freeze** Click the Freeze icon or the Thaw icon.

**New VP Freeze** Click the Freeze icon or the Thaw icon.

**VP Color** Click the [...] button to display the Select Color dialog box (page 251).

**VP Linetype** Click the [...] button to display the Select Linetype dialog box (page 706).

**VP Lineweight** Click the [...] button to display the Lineweight dialog box (page 707).

**VP Plot Style** Click the [...] button to display the Select Plot Style dialog box (page 1078).

#### Filter Preview

Displays the results of the filter as you define it. The filter preview shows which layers will be displayed in the layer list in the Layer Properties Manager when you select this filter.

## Layer Filter Examples

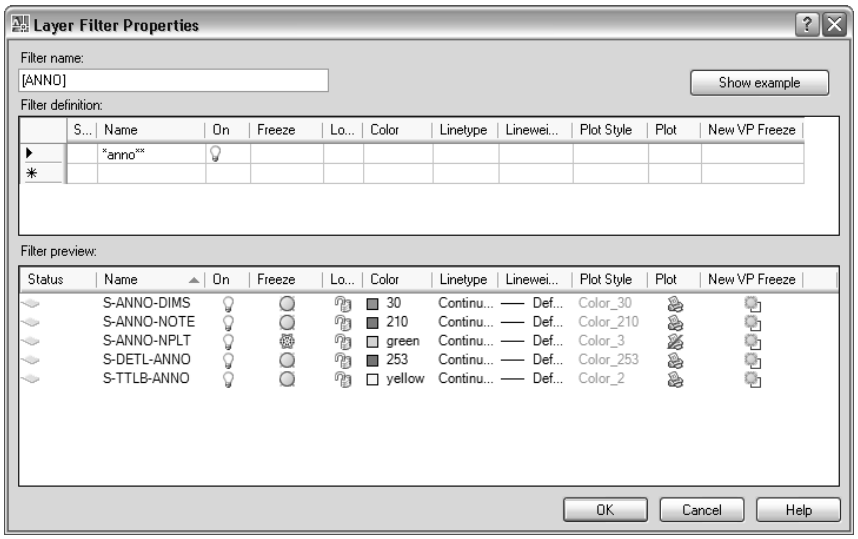
### Quick Reference

Each example shows a layer filter definition that is set up in the Layer Filter Properties dialog box.

#### Example 1

The filter named "[ANNO]" displays layers that meet *all* of the following criteria:

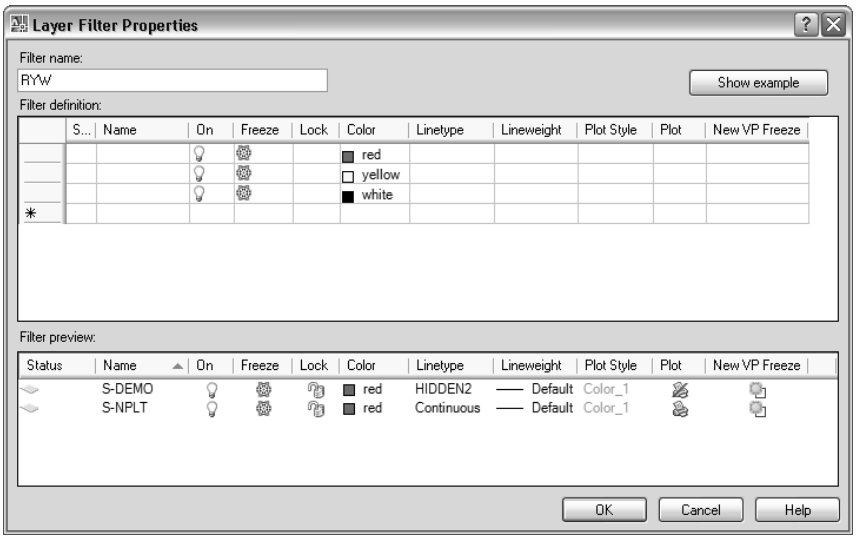
- Are in use
- Have a name that contains the letters *anno*
- Are on




### Example 2

The filter named "RYW" displays layers that meet *all* of the following criteria:

- Are on
- Are frozen
- Are red *or* yellow *or* white

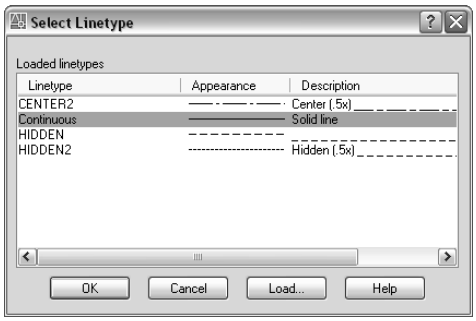



# Select Linetype Dialog Box

## Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Displays linetypes available in the current drawing.




**Loaded Linetypes** Displays the list of linetypes loaded in the current drawing.

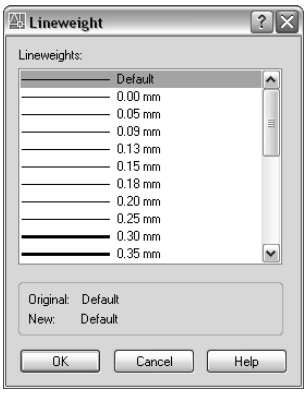
**Load** Displays the Load or Reload Linetypes dialog box (page 758), in which you can load selected linetypes into the drawing and add them to the list of linetypes.

# Lineweight Dialog Box

## Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Displays the lineweight values available. You can customize your lineweights to plot at other values by using the Plot Style Table Editor. See "Control Plotted Lineweight and Linetype" in the *User's Guide*.




**Lineweights** Displays the available lineweights to be applied. Available lineweights consist of fixed values that are most commonly used in drawings. Select a lineweight to apply it.

**Original** Displays the previous lineweight. The default lineweight assigned when a layer is created is DEFAULT (which plots to a default value of 0.01 inches or 0.25 mm). (*LWDEFAULT* system variable)

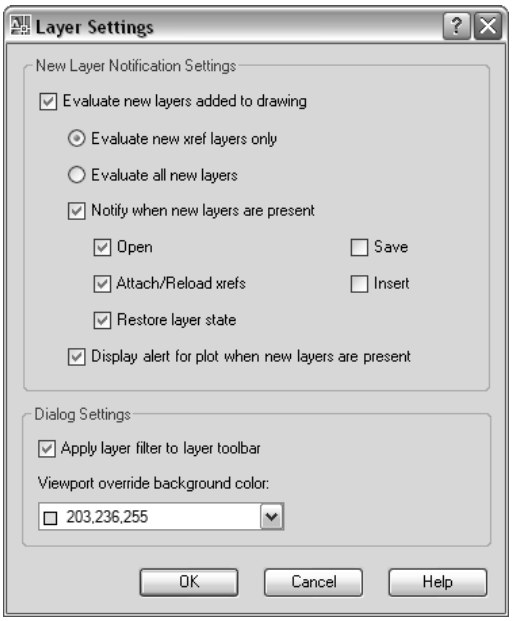
**New** Displays the new lineweight assigned to the layer.

# Layer Settings Dialog Box

## Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Controls when notification occurs for new layers and if layer filters are applied to the Layers toolbar; also controls the background color of viewport overrides in the Layer Properties Manager.




### New Layer Notification Settings

**Evaluate New Layers Added to Drawing** Checks for new layers that have been added to the drawing. (LAYEREVAL=0)

**Evaluate New Xref Layers Only** Checks for new layers that have been added to attached xrefs. (LAYEREVAL=1)

**Evaluate All New Layers** Checks for new layers that have been added to the drawing, including new layers added to attached xrefs. (LAYEREVAL=2)

**Notify when New Layers Are Present** Turns on new layer notification. (LAYERNOTIFY=0)

**Open** Displays new layer notification when new layers are present when you use the OPEN command. (LAYERNOTIFY=2)

**Attach/Reload Xrefs** Displays new layer notification when new layers are present when you are attaching or reloading xrefs. (LAYERNOTIFY=4)

**Restore Layer State** Displays new layer notification when you are restoring layer states. (LAYERNOTIFY=8)

**Save** Displays new layer notification when new layers are present when you use the SAVE command. (LAYERNOTIFY=16)

**Insert** Displays new layer notification when new layers are present when you use the INSERT command. (LAYERNOTIFY=32)

**Display Alert for Plot When New Layers are Present** Displays new layer notification when new layers are present when you use the PLOT command. (LAYERNOTIFY=1)

### Dialog Settings

**Apply Layer Filter to Layer Toolbar** Controls the display of layers in the list of layers on the Layers toolbar and Layers control panel on the dashboard by applying the current layer filter.

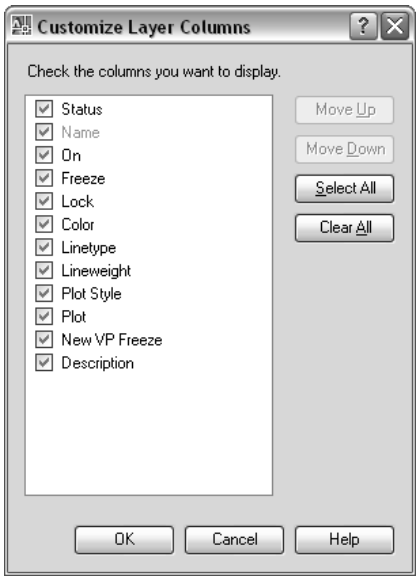
**Viewport Override Background Color** Displays a list of colors and the Select Color dialog box (page 251), where you can select a background color for viewport overrides.

## Customize Layer Columns Dialog Box

### Quick Reference

Layers
Format ➤ LayerAt the Command prompt, enter layer.
**layer** (or '**layer** for transparent use)

Controls the display and order of columns in the Layer Properties Manager.




**Check the Column to Display** Lists all columns that display depending on whether the Layer Properties Manager was accessed from Model space or a layout tab. Checked columns are displayed in the Layer Properties Manager.

**Move Up** Moves the checked layer to the left in the Layer Properties Manager.

**Move Down** Moves the checked layer to the right in the Layer Properties Manager.

**Select All** Checks all columns in the list.

**Clear All** Unchecks all columns in the list.

# -LAYER

## Quick Reference

If you enter **-layer** at the command prompt, the following LAYER command prompts are displayed.

Current layer: <"*current*">
Enter an option [? (page 711)/Make (page 711)/Set (page 711)/New (page 711)/ON (page 711)/OFF (page 712)/Color (page 712)/Ltype (page 712)/LWeight (page

713)/MATerial (page 713)/Plot (page 713)/PStyle (page 714)/Freeze (page 714)/Thaw (page 714)/LOck (page 714)/Unlock (page 714)/stAte (page 715)]:

**NOTE** The Pstyle option is available only when you are using named plot styles.

### ?—List Layers

Displays a list of the currently defined layers, showing their names, states, color numbers, linetypes, lineweights, and whether they are externally dependent layers.

Enter layer name(s) to list <*>: *Enter a name list, or press* ENTER *to list all layers*

### Make

Creates a layer and makes it current. New objects are drawn on the current layer.

Enter name for new layer (becomes the current layer) <*current*>: *Enter a name or press* ENTER

If no layer exists for the name you enter, a new layer with that name is created. The new layer is on and assumes the following properties by default: color number 7, the CONTINUOUS linetype, and a lineweight of DEFAULT.

If the layer exists but is turned off, it is turned on.

### Set

Specifies a new current layer but does not create the layer if it does not already exist. If the layer exists but is turned off, it is turned on and made current. A frozen layer cannot be made current.

Enter a layer name to make current or <select object>: *Enter a name or press* ENTER *and select an object*

### New

Creates layers. You can create two or more layers by entering names separated by commas.

Enter name list for new layer(s):

### On

Makes selected layers visible and available for plotting.

Enter name list of layer(s) to turn On:

### Off

Makes selected layers invisible and excludes them from plotting.

Enter name list of layer(s) to turn Off or <select objects>: *Enter a name list or press* ENTER *and select objects*

### Color

Changes the color associated with a layer.

Enter color name or number (1-255)[Truecolor/COlorbook]: *Enter a color name or a number from 1 through 255, enter* **t**, *or enter* **co**
**True Color** Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

**Color Book** Specifies a color from a loaded color book to used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE®**

If you enter a color book name, the following prompt is displayed:

Enter color name: *Enter the name of a color included in the selected color book, such as* **PANTONE® 573**

Enter name list of layer(s) for color current <*current*>: *Enter a name or a list of names separated by commas, or press* ENTER

The color is assigned to the layer or layers, and the layers are turned on. To assign a color but turn off the layer, precede the color with a minus sign (-).

### Ltype

Changes the linetype associated with a layer.

Enter a loaded linetype name or [?] <CONTINUOUS>: *Enter a currently loaded linetype name, enter* **?**, *or press* ENTER

If you enter a linetype or press ENTER, the following prompt is displayed:

Enter name list of layer(s) for linetype "*current*" <*current*>: *Enter a wild-card pattern, a name, or a list of names separated by commas, or press* ENTER

If you enter **?** at the Enter a Loaded Linetype Name prompt, the following prompt is displayed:

Enter linetype name(s) to list <*>: *Enter a wild-card pattern, or press* ENTER *to list all names in the drawing*

### Lweight

Changes the lineweight associated with a layer.

Enter lineweight (0.0mm-2.11mm):

If you enter a valid lineweight, the current lineweight is set to the new value. If you enter a lineweight that is not valid, the current lineweight is set to the nearest fixed lineweight value. If you would like to plot an object with a custom width not found in the list of fixed lineweight values, you can use the Plot Style Table Editor to customize plotted lineweights.

Enter name list of layers(s) for lineweight *current* <*current*>: *Enter a name list or press* ENTER

The lineweight is assigned to the layer or layers.

### Material

Attaches a material to a layer. The material must be available in the drawing before it can be assigned to a layer.

Enter material name or [?] <GLOBAL>: *Enter the name of a material currently available in the drawing, enter* **?**, *or press* ENTER

If you enter **?**, the following prompt is displayed:

Enter material name(s) to list <*>: *Press* ENTER *to list all materials in the drawing*

Objects created on the layer are assigned the material.

### Plot

Controls whether visible layers are plotted. If a layer is set to plot but is currently frozen or turned off, the layer is not plotted.

Enter a plotting preference [Plot/No plot] <Plot>: *Enter an option or press* ENTER
Enter layer name(s) for this plot preference <*current*>: *Enter a name list or press* ENTER

The plot setting is assigned to the layer or layers.

### Pstyle

Sets the plot style assigned to a layer. This option is not available if you are using color-dependent plot styles in the current drawing (the *PSTYLEPOLICY* system variable is set to 1). See "Use Plot Styles to Control Plotted Objects" in the *User's Guide*.

Enter plot style or [?] <Normal>: *Enter a name, enter* **?** *to list existing plot styles, or press* ENTER

If you select a plot style other than NORMAL, the following prompt is displayed:

Enter name list of layer(s) for plot style *current* <*current*>: *Enter the names of the layers to use this plot style, or press* ENTER *to apply the style to the current layer only*

### Freeze

Freezes layers, making them invisible and excluding them from regeneration and plotting.

Enter name list of layer(s) to freeze or <select objects>: *Enter a name list or press* ENTER *and select objects*

### Thaw

Thaws frozen layers, making them visible and available for regeneration and plotting.

Enter a name list of layer(s) to thaw:

### Lock

Locks layers, preventing editing of objects on those layers.

Enter a name list of layer(s) to lock or <select objects>: *Enter a name list or press* ENTER *and select objects*

### Unlock

Unlocks selected locked layers, permitting editing of objects on those layers.

Enter a name list of layer(s) to unlock or <select objects>: *Enter a name list or press* ENTER *and select objects*

### State

Saves and restores the state and property settings of the layers in a drawing.

Enter an option [?/Save/Restore/Edit/Name/Delete/Import/EXport]:
**?—List Named Layer States** Lists the named layer state (LAS) files in the support path for the drawing.

**Save** Saves the state and properties settings of the layers in a drawing under a specified layer state name. When saving a layer state, you specify which layer settings are affected when the layer state is later restored.

Enter new layer state name: *Enter a name and press* ENTER

Enter states to change

[On/Frozen/Lock/Plot/Newvpfreeze/Color/lineType/lineWeight/plotStyle]: *Enter the settings that you want to save and then press* ENTER

**Restore** Restores the state and property settings of all layers to previously saved settings. Restores only those layer state and property settings that were selected when the layer state was saved.

Enter name of layer state to restore or [?]: *Enter the layer state name or enter* **?** *to see a list of saved layer state names*

**Edit** Changes the saved layer settings for a specified layer state. When the layer state is restored, the specified settings are used.

Enter name of layer state to edit or [?]: *Enter the layer state name or enter* **?** *to see a list of saved layer state names*

Enter states to change

[On/Frozen/Lock/Plot/Newvpfreeze/Color/lineType/lineWeight/plotStyle]: *Enter the settings that you want to change and then press* ENTER

**Name** Changes the name of a saved layer state.

Enter name of layer state to rename or [?]: *Enter the layer state name or enter* **?** *to see a list of saved layer state names*

Enter new layer state name:

**Delete** Removes a saved layer state.

Enter name of layer state to delete or [?]: *Enter the layer state name or enter* **?** *to see a list of saved layer state names*

**Import** Loads a previously exported layer state (LAS) file, or layers states from a file (DWG, DWS, DWT) into the current drawing. Additional layers may be created as a result of importing a layer state file.

Enter file name to import <current>:

**Export** Saves a selected named layer state to a layer state (LAS) file.

Enter name of layer state to export or [?]: *Enter the layer state name or enter* **?** *to see a list of saved layer state names*

Export state to file name:

# LAYERP

## Quick Reference

Undoes the last change or set of changes made to layer settings

Layers 
**layerp**

Undoes changes you have made to layer settings such as color or linetype. If settings are restored, a message is displayed: "Restored previous layer states."

LAYERP (Layer Previous) does not undo the following changes:

- *Renamed layers:* If you rename a layer and change its properties, Layer Previous restores the original properties but not the original name.
- *Deleted layers:* If you delete or purge a layer, using Layer Previous does not restore it.
- *Added layers:* If you add a new layer to a drawing, using Layer Previous does not remove it.

# LAYERPMODE

## Quick Reference

Turns the tracking of changes made to layer settings on and off

**layerpmode**
Enter LAYERP mode [ON/OFF] <ON>: *Enter* **on** *or* **off***, or press* ENTER

When Layer Previous mode is on, changes made to layers are tracked. When Layer Previous mode is off, changes made to layers are no longer tracked.

# LAYERSTATE

### Quick Reference

Saves, restores, and manages named layer states

Layers
Format ➤ Layer States ManagerDoes not exist in the menus.
**layerstate**

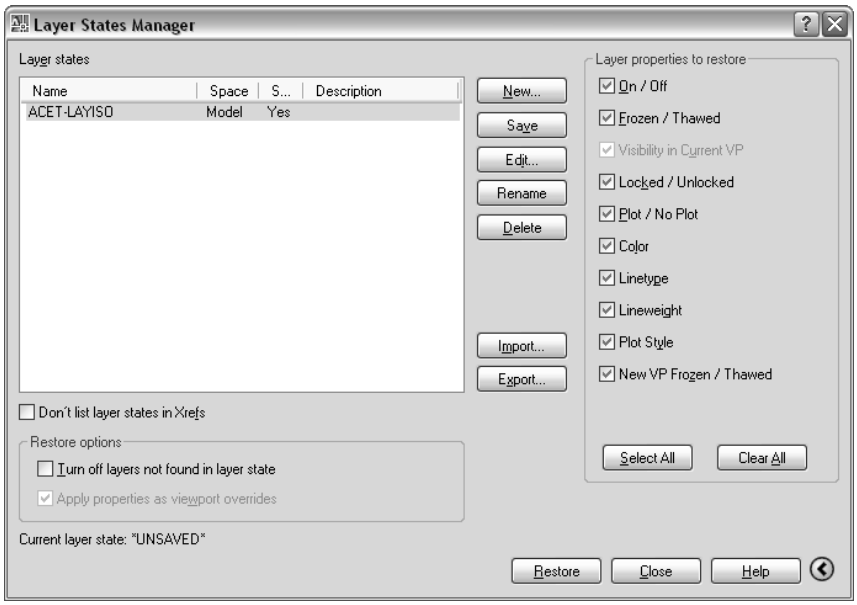
The Layer States Manager (page 717) is displayed.

## Layer States Manager

### Quick Reference

Layers
Format ➤ Layer States ManagerDoes not exist in the menus.
**layerstate**

Displays a list of saved layer states in the drawing. You can create, rename, edit, and delete layer states.




**Layer States** Lists the named layer states that have been saved in the drawing, the space in which they were saved (model space, layout, or xref), whether the layer list is the same as in the drawing, and an optional description.

**Don't List Layer States in Xrefs** Controls whether layer states in xrefs are displayed.

**New** Displays the New Layer State to Save dialog box (page 720), where you can provide a name and a description for the new named layer state.

**Save** Saves the selected named layer state.

**Edit** Displays the Edit Layer State dialog box (page 720), where you can modify a selected named layer state.

**Rename** Allows in-place editing of the layer state name.

**Delete** Removes the selected named layer state.

**Import** Displays a standard file selection dialog box, where you can load a previously exported layer state (LAS) file into the current drawing. Layer states in files (DWG, DWS, or DWT) can be imported. Additional layers may be created as a result of importing a layer state file. When a DWG, DWS, or DWT file is selected, the Select Layer States dialog box (page 722) is displayed where you can select layer states to import.

**Export** Displays a standard file selection dialog box, where you can save the selected named layer state to a layer state (LAS) file.

**Restore** Restores state and property settings of all layers in a drawing to their previously saved settings. Restores only those layer state and property settings that you specify with the check boxes.

**Close** Closes the Layer States Manager and saves changes.

### Restore Options

**Turn Off Layers Not Found in Layer State** When a layer state is restored, turns off new layers for which settings were not saved so that the drawing looks the same way it did when the named layer state was saved.

**Apply Properties as Viewport Overrides** Applies layer property overrides to the current viewport. This option is available when the Layer States Manager is accessed when a layout viewport is active.

### More Options 

Controls the display of additional options in the Layer States Manager dialog box.

- Layer Properties to Restore

### Layer Properties to Restore

Specifies the layer state settings and layer properties that are to be restored when the selected named layer state is restored. When you save a layer state on the Model tab, the Visibility in Current VP check box is not available.

**Select All** Selects all the settings.

**Clear All** Removes selection from all the settings.

## New Layer State to Save Dialog Box

### Quick Reference


Layers

Format ➤ Layer States ManagerDoes not exist in the menus.

**layerstate**

Specifies a name and a description for the layer state defined in the Layer States Manager.


**New Layer State Name** Specifies a name for the new layer state to be saved.

**Description** (Optional) Provides a description of the layer state that is displayed in the Layer States Manager.

## Edit Layer State Dialog Box

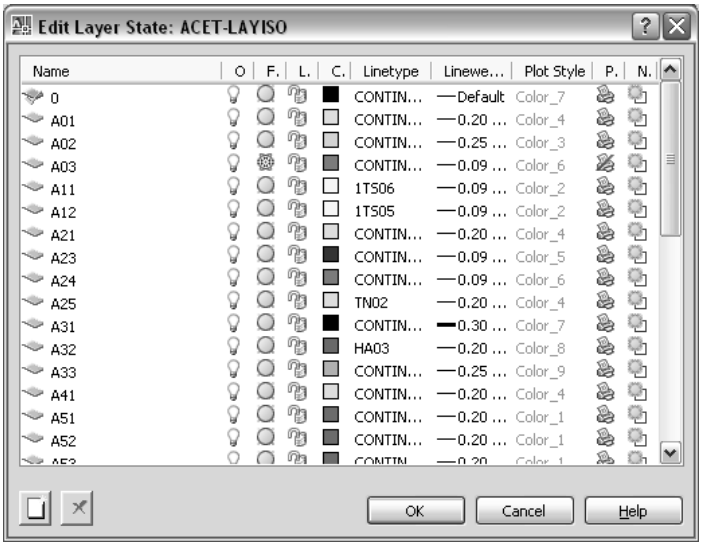
### Quick Reference


Layers

Format ➤ Layer States ManagerDoes not exist in the menus.

**layerstate**

Modifies the selected layer state. Displays all the layers and their properties that are saved in the selected layer state except for viewport override properties.




**Layer List** Displays a list of layers in the selected layer state and their properties.

**Delete** Removes the selected layers from the layer state.

**Add** Displays the Select Layers to Add to Layer States dialog box (page 721), in which you can add layers from the current drawing that are not defined in the layer state.

**Delete** Removes the selected layers from the layer state.

# Select Layers to Add to Layer State Dialog Box

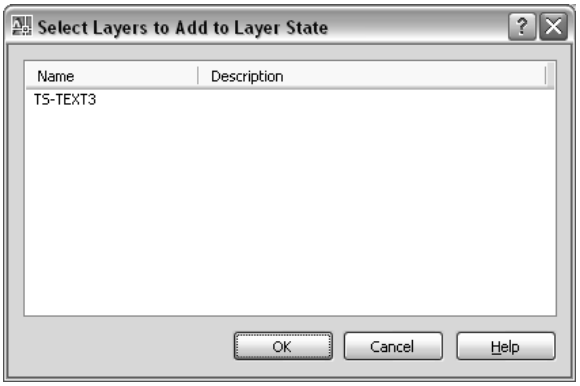
## Quick Reference

Layers

Format ➤ Layer States ManagerDoes not exist in the menus.

**layerstate**

Displays layers by name and includes descriptions that are not defined in the selected named layer state.




Multiple layers can be selected. Clicking OK displays the selected layers in the Edit Layer State dialog box (page 720).

**Layer List** Displays layers by name and description.

## Select Layer States Dialog Box

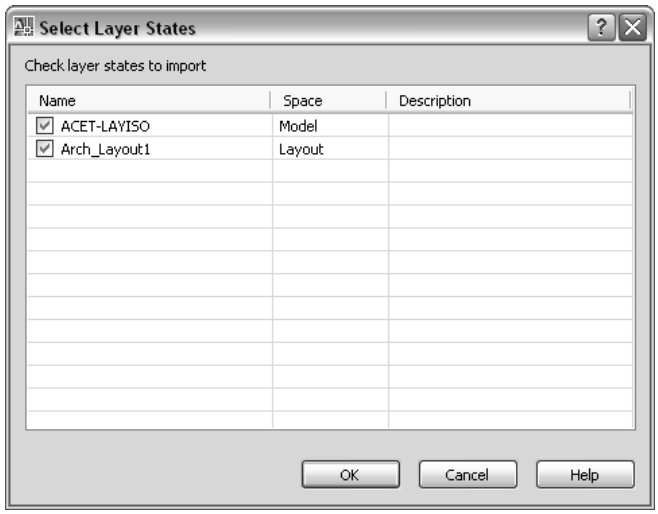
### Quick Reference

Layers

Format ➤ Layer States ManagerDoes not exist in the menus.

**layerstate**

Displays a list of layer states to be imported from a drawing file into the current drawing.




**Check Layer States to Import** Displays a list of layer states, the space (model tab or layout tab) in which the layer state was saved, and a description.

The layer state names in the selected file (DWG, DWS, DWT) are compared to the layer state names in the current drawing. Duplicate layer state names are not checked by default. When a duplicate layer state name is checked, a prompt displays if you want to overwrite the existing layer state.

---

**NOTE** If the layer state to be imported contains linetypes or plot styles that do not exist in the current drawing, they are imported from the DWG, DWS or DWT file.

---

# LAYFRZ

## Quick Reference

Freezes the layer of selected objects


Layers II

Format ➤ Layer Tools ➤ Layer FreezeDoes not exist in the menus.

**layfrz**

Current settings: Viewports=Vpfreeze, Block nesting level=Block (Default settings value)
Select an object on a layer to be frozen (page 724) or [Settings (page 724)/Undo (page 724)]: *Select an object or enter* **s** *or* **u**
Layer <layer name> has been frozen.

### Select an Object on a Layer to be Frozen

Specifies the layer to be frozen.

Layer "<*layer name*>" has been frozen.

### Settings

Displays settings for viewports and block definitions. The setting you choose persists from session to session.

Enter setting type for [Viewports/Block selection]:
**Viewports** Displays settings for viewports.

In paper space viewport use [Freeze/Vpfreeze] <Vpfreeze>: *Enter* **f** *to freeze objects in all viewports or* **v** *to freeze an object in the current viewport only*

**Block Selection** Displays settings for block definitions.

Enter Block Selection nesting level [Block/Entity/None] <Block>: *Enter* **b**, **e**, *or* **n** *to set the default*

**Block** If a selected object is nested in a block, freezes the layer of that block. If a selected object is nested in an xref, freezes the layer of the object.

**Entity** Freezes the layers of selected objects even if they are nested in an xref or a block.

**None** If a block or an xref is selected, freezes the layer containing that block or xref.

### Undo

Cancels the previous layer selection.

# LAYISO

## Quick Reference

Hides or locks all layers other than those of the selected objects

Layers II 
Format ➤ Layer Tools ➤ Layer Isolate
Layer panel (click icon to expand), Layer Isolate
**layiso**
Current setting: <*current settings*>
Select objects on the layer(s) to be isolated or [Settings]: *Select objects or enter* **s**

### Select Objects on the Layer(s) to be Isolated

After selecting one or more objects, all layers *except* the layers of the selected objects are either turned off, frozen in the current layout viewport, or locked, depending on the current setting. The layers that remain visible and unlocked are termed *isolated*.

---

**NOTE** Locked layers are dimmed by default. You can specify the percent of the dimming from the Lock option in this command. You can later change the value with the *LAYLOCKFADECTL* system variable.

---

If you make changes to layers within a session and you want to restore the layers to the state they were in immediately before you entered the LAYISO command, use the *LAYUNISO* command.

### Settings

Controls whether layers are turned off, frozen in the current layout viewports, or locked.

Enter setting for layers not isolated [Off/Lock and fade] <Lock>: *Enter an option*

### Off

Turns off or freezes all layers *except* the layers of the selected objects.

In paper space viewport use [Vpfreeze/Off] <Vpfreeze>: *Enter an option*

**Vpfreeze** In a layout, freezes all but the selected layers in the current layout viewport only. Other layout viewports in the drawing are unchanged.

If not in a layout, all other layers are turned off instead.

**Off** Turns off all but the selected layers in all viewports.

#### Lock

Locks all layers *except* the layers of the selected objects, and sets the dimming for locked layers.

Enter fade value (0-90) <50>: *Enter a number for the percent dimming*
Select objects on the layer(s) to be isolated or [Settings]: *Select objects or enter* **s**

# LAYLCK

## Quick Reference

Locks the layers of selected objects

Layers II 
Format ➤ Layer Tools ➤ Layer LockDoes not exist in the menus.
**laylck**
Select an object on the layer to be locked:

# LAYMCH

## Quick Reference

Changes the layer of a selected object to match the destination layer.

Layers II 
Format ➤ Layer Tools ➤ Layer MatchDoes not exist in the menus.
**laymch**
Select objects to be changed: *Select an object to change its layer*

Select object on destination layer (page 727) or [Name (page 727)]: *Select an object or enter* **n** *to open the Change to Layer dialog box*

If you enter **-laymch** at the command prompt, options are displayed at the command prompt (page 727).

#### Select Object on Destination Layer

Select an object on the destination layer.

#### Name

Displays the Change to Layer dialog box (page 727).

## Change to Layer Dialog Box

### Quick Reference

Layers II
Format ➤ Layer Tools ➤ Change to Current Layer
**laymch**

Displays a list of layers to choose as the destination layer. You can also create a new destination layer.

**Current Layer** Displays the currently selected layer. Create a new layer name by entering a name.

**Destination Layer list** Displays a list of layers in the drawing.

## -LAYMCH

### Quick Reference

If you enter **-laymch** at the command prompt, the following LAYMCH command prompts are displayed.

Select objects to be changed:
Select objects: *Select an object to change its layer*

After you select the object(s) to change, the following prompt is displayed:

Select object on destination layer (page 728) or [Name (page 728)]: *Select an object or enter* **n** *to enter a layer name*

#### Select Object on Destination Layer

Select an object on the destination layer. *Select an object whose layer you want to match*
N object(s) changed to layer <*layer name*>

#### Name

Enter layer name: *Enter the name of a layer you want the selected objects to match*
N object(s) changed to layer <*layer name*>

# LAYMCUR

## Quick Reference

Makes the layer of a selected object current

Layers 
Format ➤ Layer Tools ➤ Make Object's Layer Current
**laymcur**
Select object whose layer will become current: *Select an object to make its layer current*

# LAYMRG

## Quick Reference

Merges selected layers onto a destination layer

Format ➤ Layer Tools ➤ Layer MergeDoes not exist in the menus.
**laymrg**
Select object on layer to merge (page 729)or [Name (page 729)]: *Select an object or enter* **n** *to select a layer in the Merge Layers dialog box*

After you selected the layer(s) to merge, the following prompt is displayed:

Select object on target layer (page 729)or [Name (page 729)]: *Select an object or enter* **n** to *select a layer in the Merge to Layer dialog box*

The layer you selected to merge into the target layer is deleted.

If you enter **-laymrg** at the command prompt, options are displayed at the command prompt (page 730).

#### Select Object on Layer to Merge

Select an object on the layer you want to merge.

Selected layers: *<layer name>*
Select object on layer to merge or [Name/Undo]: *Select an object, enter* **n** *to display the Merge Layers dialog box, or enter* **u**

#### Name

Displays the Merge Layers dialog box (page 729).

#### Select Object on Target Layer

Select an object on the destination layer.

Select object on the target layer or [Name]: *Select an object, or enter* **n** *to display the Merge to Layer dialog box*

#### Name

Displays the Merge to Layer dialog box (page 730).

## Merge Layers Dialog Box

### Quick Reference

Format ➤ Layer Tools ➤ Layer MergeAt the Command prompt, enter laymrg.
**laymrg**
**Merge Layer list** Displays a list of layers to merge.

## Merge to Layer Dialog Box

### Quick Reference

Format ➤ Layer Tools ➤ Layer MergeAt the Command prompt, enter laymrg.
**laymrg**

Displays a list of layers onto which the selected layers or objects can be merged.

**Target Layer** Displays the first layer in an alphabetical list of the layers in the current drawing.

**Target Layer list** Displays an alphabetical list of the layers onto which you can merge the selected layers or objects.

## -LAYMRG

### Quick Reference

If you enter **-laymrg** at the command prompt, the following LAYMRG command prompts are displayed.

Select object on layer to merge (page 730)or [Name (page 730)]: *Select an object or enter* **n** to *select a layer from a list of layers*

After you selected the layer(s) to merge, the following prompt is displayed:

Select object on target layer (page 731)or [Name (page 731)]: *Select an object or enter* **n** to *select a target layer from a list of layers*

The layer you selected to merge into the target layer is deleted.

#### Select Object on Layer to Merge

Select an object on the destination layer.

Selected layers: <*layer name*>
Select object on layer to merge or [Name]: *Select an object or enter* **n**

#### Name

Displays a list of layers, where you can select layers to merge.

Select a layer to merge.

Enter layer name or [?]: *Enter a layer name or enter* **?**
Enter layer name(s) to list <*>: * *Enter a layer name or press ENTER to list all layers.*

#### Select Object on Target Layer

Select an object on the target layer.

Selected layers: *<layer name>*
Select object on layer to merge or [Name]: *Select an object, enter* **n** *to display a list of layers on*

#### Name

Displays a list of layers onto which you can merge the selected object or layer.

Enter layer name or [?]: *Enter a layer name or enter* **?**
Enter layer name(s) to list <*>: * *Enter a layer name or press ENTER to list all layers.*

# LAYOFF

## Quick Reference

Turns off the layer of the selected object

Layers II 
Format ➤ Layer Tools ➤ Layer OffDoes not exist in the menus.
**layoff**
*Current settings*: Viewports= , Block nesting level=
Select an object on the layer to be turned off (page 731)or [Settings (page 731)/Undo (page 732)]: *Select an object, enter* **s**, *or enter* **u**

#### Select an Object on the Layer to be Turned Off

Selects one or more objects whose layers you want to turn off.

#### Settings

Displays the Viewports and Block Definition setting types. The setting you choose persists from session to session.

Enter setting type for [Viewports/Block selection]:

**Viewports** Displays the Viewports setting types.

Returns the following prompt:

In paper space viewport use [Vpfreeze/Off] <Vpfreeze>: *Enter* **v** *to freeze the selected layer in the current viewport, or enter* **o** *to turn off selected layers in all viewports*

**Vpfreeze** In paper space, freezes the layer selected in the current viewport.

**Off** In paper space, turns off selected layers in all viewports.

**Block Selection** Displays the Block Selection setting types, where you can freeze layers of selected objects.

Enter Block Selection nesting level [Block/Entity/None] <Block>: *Enter* **b**, **e,** *or* **n** *to set the default option*

- **Block**: Turns off the layers of selected objects. If a selected object is nested in a block, the layer containing that block is turned off. If a selected object is nested in an xref, the layer of the object is turned off.
- **Entity**: Turns off layers of selected objects even if they are nested in an xref or a block.
- **None**: Turns off the layers of selected objects. If a block or an xref is selected, the layer containing that block or xref is turned off.

#### Undo

Cancels the previous layer selection.

# LAYON

## Quick Reference

Turns on all layers

Format ➤ Layer Tools ➤ Turn All Layers OnDoes not exist in the menus.
**layon**
All layers in the drawing are turned on.

# LAYOUT

## Quick Reference

Creates and modifies drawing layout tabs

Layouts 
Insert ➤ Layout
**layout**
Enter layout option [Copy (page 733)/Delete (page 733)/New (page 733)/Template (page 733)/Rename (page 734)/SAveas (page 734)/Set (page 734)/? (page 734)] <set>:

**NOTE** Many of these options are available by right-clicking a layout tab name.

**Copy** Copies a layout. If you do not provide a name, the new layout assumes the name of the copied layout with an incremental number in parentheses. The new tab is inserted before the copied layout tab.

Enter name of layout to copy <*current*>:

Enter layout name for copy <*default*>:

**Delete** Deletes a layout. The most current layout is the default.

Enter name of layout to delete <*current*>:

The Model tab cannot be deleted. To remove all the geometry from the Model tab, you must select all geometry and use the *ERASE* command.

**New** Creates a new layout tab. Up to 255 layouts can be created in a single drawing.

Enter name of new layout <Layout#>:

Layout names must be unique. Layout names can be up to 255 characters long and are not case sensitive. Only the first 31 characters are displayed on the tab.

**Template** Creates a new layout tab based on an existing layout in a template (DWT), drawing (DWG), or drawing interchange (DXF) file. If the *FILEDIA* system variable is set to 1, a standard file selection dialog box (page 931) is displayed for selecting a DWT, DWG, or DXF file. After you select a file, the Insert Layouts dialog box is displayed, which lists the layouts saved in the selected file. After you select a layout, the layout and all objects from the specified template or drawing file are inserted into the current drawing.

**Rename** Renames a layout. The last current layout is used as the default for the layout to rename.

Enter layout to rename <*current*>:

Enter new layout name <*current*>:

Layout names must be unique. Layout names can be up to 255 characters long and are not case sensitive. Only the first 31 characters are displayed on the tab.

**Saveas** Saves a layout as a drawing template(DWT) file without saving any unreferenced symbol table and block definition information. You can then use the template to create new layouts in your drawings without having to eliminate unnecessary information. See "Reuse Layouts and Layout Settings" in the *User's Guide*.

Enter layout to save to template <*current*>:

The last current layout is used as the default for the layout to save as a template. If the *FILEDIA* system variable is set to 1, a standard file selection dialog box is displayed in which you can specify the template file in which to save the layout. The default layout template directory is specified in the Options dialog box (page 946).

**Set** Makes a layout current.

Enter layout to make current <last>:

**?—List Layouts** Lists all the layouts defined in the drawing.

# LAYOUTWIZARD

## Quick Reference

Creates a new layout tab and specifies page and plot settings

Insert ➤ Layout ➤ Create Layout WizardDoes not exist in the menus.
**layoutwizard**

The Layout wizard (page 735) is displayed.

## Layout Wizard

### Quick Reference

**layoutwizard**

The Layout wizard contains a series of pages that step you through the process of creating a new layout. You can choose to create a new layout from scratch or use an existing layout template on which to base your new layout.

Depending on which plotting device is currently configured, you can select a paper size from those available. You can select a predefined title block to apply to your new layout. The Preview area displays a preview image of the title block you select.

When you've finished using the wizard to specify your layout settings, you can modify any of the settings using the *PAGESETUP* command from within the new layout.

# LAYTHW

### Quick Reference

Thaws all layers

Format ➤ Layer tools ➤ Thaw All Layers.Does not exist in the menus
**laythw**
All layers in the drawing are thawed.

**NOTE** LAYTHW does not thaw layers in a viewport. Use the VPLAYER (page 1529) command to thaw layers in a viewport.

# LAYTRANS

### Quick Reference

Changes a drawing's layers to layer standards you specify

CAD Standards 
Tools ➤ CAD Standards ➤ Layer TranslatorAt the Command prompt, enter laytrans.
**laytrans**

The Layer Translator (page 736) is displayed.

# Layer Translator

## Quick Reference

CAD Standards
Tools ➤ CAD Standards ➤ Layer TranslatorAt the Command prompt, enter laytrans.
**laytrans**

In the Layer Translator, you specify the layers in the current drawing that you want to translate, and the layers to translate them to.

### Translate From

Specifies the layers to be translated in the current drawing. You can specify layers by selecting layers in the Translate From list or by supplying a selection filter.


The color of the icon preceding the layer name indicates whether or not the layer is referenced in the drawing. A dark icon indicates that the layer is referenced; a white icon indicates the layer is unreferenced. Unreferenced layers can be deleted from the drawing by right-clicking in the Translate From list and choosing Purge Layers.

**Selection Filter** Specifies layers to be selected in the Translate From list, using a naming pattern that can include wild-cards. For a list of valid wild-cards, see the table in "Filter and Sort the List of Layers" in the *User's Guide*. The layers identified by the selection filter are selected in addition to any layers previously selected.

**Select** Selects the layers specified in Selection Filter.

### Map

Maps the layers selected in Translate From to the layer selected in Translate To.

### Map Same

Maps all layers that have the same name in both lists.

### Translate To

Lists the layers you can translate the current drawing's layers to.

**Load** Loads layers in the Translate To list using a drawing, drawing template, or standards file that you specify. If the specified file contains saved layer mappings, those mappings are applied to the layers in the Translate From list and are displayed in Layer Translation Mappings.

You can load layers from more than one file. If you load a file that contains layers of the same name as layers already loaded, the original layers are retained and the duplicate layers are ignored. Similarly, if you load a file containing mappings that duplicate mappings already loaded, the original mappings are retained and the duplicates are ignored.

**New** Defines a new layer to be shown in the Translate To list for translation. If you select a Translate To layer before choosing New, the selected layer's properties are used as defaults for the new layer. You cannot create a new layer with the same name as an existing layer.

### Layer Translation Mappings

Lists each layer to be translated and the properties to which the layer will be converted. You can select layers in this list and edit their properties using Edit.

**Edit** Opens the Edit Layer dialog box (page 739), where you can edit the selected translation mapping. You can change the layer's linetype, color, and lineweight. If all drawings involved in translation use plot styles, you can also change the plot style for the mapping.

**Remove** Removes the selected translation mapping from the Layer Translation Mappings list.

**Save** Saves the current layer translation mappings to a file for later use.

Layer mappings are saved in the DWG or DWS file format. You can replace an existing file or create a new file. The Layer Translator creates the referenced layers in the file and stores the layer mappings in each layer. All linetypes used by those layers are also copied into the file.

#### Settings

Opens the Settings dialog box (page 738), where you can customize the process of layer translation.

#### Translate

Starts layer translation of the layers you have mapped.

If you have not saved the current layer translation mappings, you are prompted to save the mappings before translation begins.

## Settings Dialog Box

### Quick Reference

CAD Standards
Tools ➤ CAD Standards ➤ Layer TranslatorAt the Command prompt, enter laytrans.
**laytrans**

Controls the process of layer translation.

**Force Object Color to BYLAYER** Specifies whether or not every object translated takes on the color assigned to its layer. If this option is selected, every object takes on the color of its layer. If this option is cleared, every object retains its original color.

**Force Object Linetype to BYLAYER** Specifies whether or not every object translated takes on the linetype assigned to its layer. If this option is selected, every object takes on the linetype of its layer. If this option is cleared, every object retains its original linetype.

**Translate Objects in Blocks** Specifies whether or not objects nested within blocks are translated. If this option is selected, nested objects in blocks are translated. If this option is cleared, nested objects in blocks are not translated.

**Write Transaction Log** Specifies whether or not a log file detailing the results of translation is created. If this option is selected, a log file is created in the

same folder as the translated drawing. The log file is assigned the same name as the translated drawing, with a *.log* file name extension. If the Write Transaction Log option is cleared, no log file is created.

**Show Layer Contents When Selected** Specifies which layers to display in the drawing area. If this option is selected, only the layers selected in the Layer Translator dialog box are displayed in the drawing area. If this option is cleared, all layers in the drawing are displayed.

## Edit/New Layer Dialog Box

### Quick Reference

CAD Standards
Tools ➤ CAD Standards ➤ Layer TranslatorAt the Command prompt, enter laytrans.
**laytrans**

Sets or modifies layer properties.

**Name** Specifies the layer name.

**Linetype** Specifies the layer linetype.

**Color** Specifies the layer color.

**Lineweight** Specifies the layer lineweight.

**Plot Style** Specifies the layer plot style. You can modify the plot style only if all drawings referenced by the Layer Translator use named plot styles.

# LAYULK

### Quick Reference

Unlocks the layer of a selected object

Layers II
Format ➤ Layer Tools ➤ Layer UnlockDoes not exist in the menus.
**layulk**

When you hover the cursor over objects on locked layers, the locked icon is displayed.

Select an object on the layer to be unlocked:

# LAYUNISO

## Quick Reference

Turns on layers that were turned off with the last LAYISO command

Layers II
Format ➤ Layer Tools ➤ Layer UnisolateDoes not exist in the menus.
Layer panel (click icon to expand), Unisolate
**layuniso**

LAYUNISO restores layers to the state they were in just before you entered the *LAYISO* command. Changes to layer settings after LAYISO is used are retained when you enter the LAYUNISO command. If LAYISO was not used, LAYUNISO does not restore any layers.

**NOTE** You can also restore layers to their previous layer state by using the Layer Previous button on the Layers toolbar (or by entering *LAYERP* at the command prompt), as long as you have not made any changes to layer settings.

# LAYVPI

## Quick Reference

Isolates an object's layer to the current viewport

Format ➤ Layer Tools ➤ Isolate Layer to Current Viewport
**layvpi**

Isolates the layer of a selected object in the current viewport by freezing the layer in all but the current viewport. You can choose to isolate all layouts or only the current layout.

**NOTE** LAYVPI works only when TILEMODE is set to 0 and when two or more paper space viewports are defined.

*Current setting*s: Layouts= , Block nesting level=
Select objects on the layer to be isolated in viewport (page 741) or [Settings (page 741)/Undo (page 741)]: *Select objects or enter an option*

### Select Objects on the Layer to be Isolated in Viewport

Selects an object whose layer you want to isolate in a viewport.

### Settings

Displays the Viewports and Block Definition setting types. The setting you choose persists from session to session.

Enter setting type for [Viewports/Block selection]:
**Layouts** Displays layout options for isolating layers.

Isolate layers in all viewports except current for [All layouts/Current layout] *Enter All Layouts or Current Layout*

- **All Layouts**: In all layouts, isolates layers of selected objects in all but the current viewport
- **Current Layout**: In the current layout, isolates layers of selected objects in all but the current viewport. This option persists from session to session.

**Block Selection** Displays the Block Selection setting types, where you can freeze layers of selected objects.

Enter Block Selection nesting level [Block/Entity/None] <Block>: *Enter* **b**, **e**, *or* **n** *to set the default option*

- **Block**: Isolates the layers of selected objects. If a selected object is nested in a block, the layer containing that block is isolated. If a selected object is nested in an xref, the layer of the object is isolated.
- **Entity**: Isolates layers of selected objects even if they are nested in an xref or a block.
- **None**: Isolates the layers of selected objects. If a block or an xref is selected, the layer containing that block or xref is isolated.

### Undo

Cancels the previous layer selection.

# LAYWALK

## Quick Reference

Dynamically displays layers in a drawing

Layers II 
Format ➤ Layer Tools ➤ Layer WalkDoes not exist in the menus.
**laywalk**

The LayerWalk dialog box (page 742) is displayed.

# LayerWalk Dialog Box

## Quick Reference

Layers II
Format ➤ Layer Tools ➤ Layer WalkDoes not exist in the menus.
laywalk

Dynamically displays objects on layers that you select in the Layer list. The number of layers in the drawing are displayed in the dialog box title. You can change the current layer state when you exit, save layer states, and purge layers that are not referenced.

You can use the LayerWalk dialog box in a paper space viewport to select layers to turn on and thaw in the layer table and the current viewport. Any layer that is not selected in the Layer list is frozen in the current viewport. You can change the display of one viewport without altering the display of another viewport.

### Filter

Turns an active filter on and off. When this check box is selected, the list displays only those layers that match the active filter. When this check box is cleared, the full list of layers is displayed. (This option is available only when there is an active filter.) To turn on an active filter, in the filter list, either enter a wildcard and press ENTER, or select a saved filter.

### Layer List

If a filter is active, displays a list of layers defined in that filter. If no filter is active, displays a list of layers in the drawing. Double-click a layer to set it to Always Show (an asterisk to the left of the layer is displayed). Right-click in the layer list to display more options.

In the layer list, you can

- Click a layer name to display the contents of a layer.
- Double-click a layer name to turn the Always Show option on or off.
- Press CTRL and click layers to select multiple layers.
- Press SHIFT and click to select layers in a sequence.
- Press either CTRL or SHIFT and double-click in the layer list to turn the Always Show option on or off.
- Click and drag in the layer list to select multiple layers.
  For more options in the layer list, right-click to access the Layer List shortcut menu.

### Layer List Shortcut Menu Options

Displays a list of options for the layers selected in the Layer list.

**Hold Selection** Turns on the Always Show option for selected layers. An asterisk (*) is displayed to the left of each layer held.

**Release Selection** Turns off the Always Show option for selected layers.

**Release All** Turns off the Always Show option for all layers.

**Select All** Selects and displays all layers.

**Clear All** Clears all layers.

**Invert Selection** Clears current layers and selects and displays all other layers.

**Select Unreferenced** Selects all unreferenced layers. Use with the Purge button to remove unused layers.

**Save Layer State** Saves the current selection of layers as a layer state that can be used by the Layer States Manager.

**Inspect** Displays the number of layers in the drawing, the number of layers selected, and the number of objects on the selected layers.

**Copy as Filter** Displays the name of the selected layer in the Filter text box. Can be used to create wildcards.

**Save Current Filter** Saves the current filter so that it is available in the Filter list for reuse.

**Delete Current Filter** Removes the current filter from the filter list.

#### Select Objects

Selects objects and their layers.

#### Filter List

Filters the layer list. Enter a wildcard and press ENTER to display and highlight only those layers whose names match the wildcard. Right-click to save and delete filters. The filter list displays saved filters only.

For more information about wild-card characters, see the Wild-Card Characters section of Filter and Sort the List of Layers.

#### Purge

When selected layers are not referenced, purges them from the drawing. For a list of layers that can be purged, right-click anywhere in the Layer list and click Select Unreferenced. In the Layer list, the unreferenced layers are highlighted. You can purge those layers.

#### Restore on Exit

Returns layers to their previous state when you exit the dialog box. If the check box is cleared, any changes you made are saved.

# LEADER

## Quick Reference

Creates a line that connects annotation to a feature

**leader**
Specify leader start point:
Specify next point:

It is recommended that you use the workflow available through the MLEADER (page 838) command to create leader objects. For more information about multileader objects, see Create and Modify Leaders.

A leader line segment is drawn and prompts for points and options are displayed.

Specify next point (page 745) or [Annotation (page 745)/Format (page 747)/Undo (page 747)] <Annotation>: *Specify a point, enter an option, or press* ENTER

### Point Specification

Draws a leader line segment to the point specified and continues to prompt you for points and options.

Specify next point or [Annotation/Format/Undo] <Annotation>: *Specify a point, enter an option, or press* ENTER

### Annotation

Inserts an annotation at the end of the leader line. The annotation can be single or multiple lines of text, a feature control frame containing geometric tolerances, or a block.

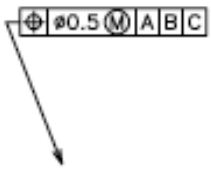
Enter first line of annotation text or <options>: *Enter text or press* ENTER

If you enter text at the Annotation prompt, the text is inserted at the end of the leader line. You are prompted for additional lines of text until you end the command by pressing ENTER twice.

If you press ENTER at the Annotation prompt without entering text first, the following prompt is displayed:

Enter an annotation option [Tolerance/Copy/Block/None/Mtext] <Mtext>: *Enter an option or press* ENTER
**Tolerance** Creates a feature control frame containing geometric tolerances using the Geometric Tolerance dialog boxes (see *TOLERANCE*).

You can create datum indicators and basic dimension notation in these dialog boxes. After you specify the geometric tolerance, LEADER ends.



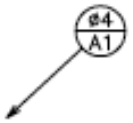
**Copy** Copies text, a multiline text object, a feature control frame with geometric tolerances, or a block and connects the copy to the end of the leader line. The copy is associated with the leader line, meaning that if the copied object moves, the end of the leader line moves with it. The display of the hook line depends on the object copied.

Select an object to copy:

The object is inserted and LEADER ends. The value of the current text gap (see *DIMSTYLE* or the *DIMGAP* system variable) determines where the text and multiline text objects are inserted. Blocks or feature control frames with geometric tolerances are attached to the end of the leader line.

**Block** Inserts a block at the end of the leader line. The prompts are the same as for INSERT (page 669). The block reference is inserted at an offset from the end of the leader line and is associated to the leader line, meaning that if the block moves, the end of the leader line moves with it. No hook line is displayed.

Enter block name or [?]: *Enter a block name or enter* ***?*** *to list all blocks in the drawing*

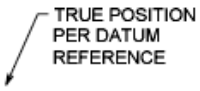



**None** Ends the command without adding any annotation to the leader line.

**Mtext** Creates text using the In-Place Text Editor (page 873) when you specify an insertion point and a second point for the text boundary.

Enter the characters for the text. Enclose format strings for prefixes and suffixes in angle brackets (< >). Enclose format strings for alternate units in square brackets ([ ]). For more information about adding a prefix or suffix, see "Control the Display of Dimension Units" in the *User's Guide*.

The units settings and the current text style determine how the text is displayed. The multiline text is vertically centered and horizontally aligned according to the *X* axis direction of the last two vertices of the leader line. The text is offset from the hook line by the distance specified under Offset from Dim Line on the Text tab of the New, Modify, or Override Dimension Style dialog box (page 448). If the offset specified is negative, the multiline text is enclosed in a box as a basic dimension.



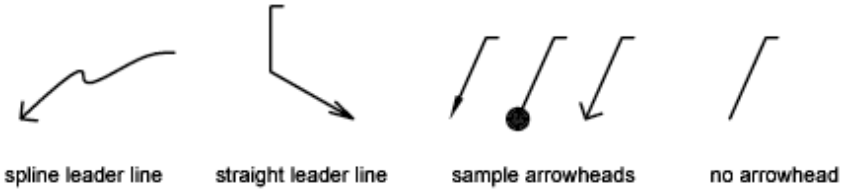
After you specify the text, LEADER ends.

### Format

Controls the way the leader is drawn and whether it has an arrowhead.

Enter leader format option [Spline/STraight/Arrow/None] <Exit>: *Enter an option or press* ENTER*to return to the previous prompt*

After each option, the Specify Next Point prompt is redisplayed.




**Spline** Draws the leader line as a spline. The vertices of the leader line are the control points, each of equal unit weight.

**Straight** Draws the leader line as a set of straight line segments.

**Arrow** Draws an arrowhead at the start point of the leader line.

**None** Draws a leader line with no arrowhead at the start point.

**Exit** Exits the Format options.

### Undo

Undoes the last vertex point on the leader line. The previous prompt is displayed.

# LENGTHEN

## Quick Reference

Changes the length of objects and the included angle of arcs

Modify ➤ LengthenAt the Command prompt, enter lengthen.
**lengthen**
Select an object (page 748) or [DElta (page 748)/Percent (page 749)/Total (page 749)/DYnamic (page 750)]: *Select one object or enter an option*

### Object Selection

Displays the length and, where applicable, the included angle of the object.

LENGTHEN does not affect closed objects. The extrusion direction of the selected object need not be parallel to the *Z* axis of the current user coordinate system (UCS).

Current length: <*current*>, included angle: <*current*>
Select an object or [DElta/Percent/Total/DYnamic]: *Select one object, enter an option, or press* ENTER *to end the command*

### Delta

Changes the length of an object by a specified increment, measured from the endpoint that is closest to the selection point. Delta also changes the angle of an arc by a specified increment, measured from the endpoint that is closest to the selection point. A positive value extends the object; a negative value trims it.

Enter delta length or [Angle] <*current*>: *Specify a distance, enter* **a**, *or press* ENTER

### Delta Length

Changes the length of the object by the specified increment.

Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.


### Angle

Changes the included angle of the selected arc by the specified angle.

Enter delta angle <*current*>: *Specify an angle or press* ENTER
Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.


### Percent

Sets the length of an object by a specified percentage of its total length.

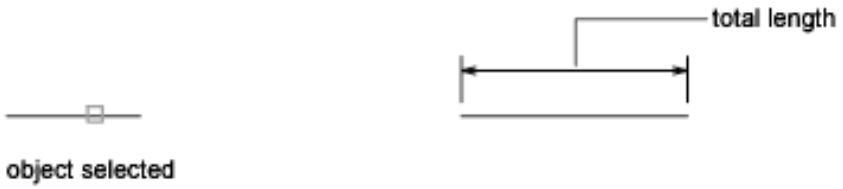
Enter percentage length <*current*>: *Enter a positive nonzero value or press* ENTER
Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

### Total

Sets the length of a selected object by specifying the total absolute length from the fixed endpoint. Total also sets the included angle of a selected arc by a specified total angle.

Specify total length or [Angle] <*current*>: *Specify a distance, enter a positive nonzero value, enter* **a**, *or press* ENTER




### Total Length

Lengthens the object to the specified value from the endpoint that is closest to the selection point.

Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

### Angle

Sets the included angle of the selected arc.

Specify total angle <*current*>: *Specify an angle or press*ENTER
Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.


**Dynamic**

Turns on Dynamic Dragging mode. You change the length of a selected object by dragging one of its endpoints. The other end remains fixed.

Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

# LIGHT

## Quick Reference

Creates a light

**light**
Enter light type [Point/Spot/Web/Targetpoint/Freespot/FreeweB/Distant] <Point>:

Depending on the type of light you specify, the prompts that are displayed are identical to the prompts in the *POINTLIGHT*, *SPOTLIGHT*, *WEBLIGHT*, *TARGETPOINT*, *FREESPOT*, *FREEWEB* or *DISTANTLIGHT* command.

# LIGHTLIST

### Quick Reference

Opens the Lights in Model window to add and modify lights

Render
View ➤ Render ➤ Light ➤ Light ListDoes not exist in the menus.
**lightlist**
Light panel, Light List

Opens the Lights in Model window (page 751), which lists the lights in the drawing.

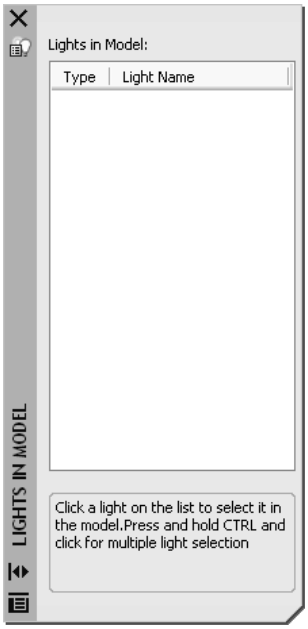
## Lights in Model Window

### Quick Reference

Render
View ➤ Render ➤ Light ➤ Light ListDoes not exist in the menus.
**lightlist**
Light panel, Light List

Lists the lights in the drawing. An icon in the Type column indicates the type of light: point light, spotlight, or distant light, and whether it is on or off.

Select a light in the list to select it in the drawing. To sort the list, click the Type or Name column head.




**Delete Light or Delete Lights** With one or more lights selected, right-click and click Delete Light or Delete Lights to remove the light from the drawing. You can also press DELETE.

**Properties** With one or more lights selected, right-click and click Properties to display the Properties palette (page 1109), where you can change the properties of lights and turn them on and off. When a property is selected, the panel at the bottom displays a description of the property. You can also double-click to display the Properties palette.

# LIGHTLISTCLOSE

## Quick Reference

Closes the Lights in Model window

**lightlistclose**
Light panel, Light List

The Lights in Model window (page 751) closes.

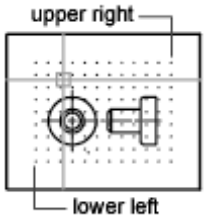
# LIMITS

## Quick Reference

Sets and controls the limits of the grid display in the current Model or layout tab

Format ➤ Drawing LimitsAt the Command prompt, enter limits.
**limits** (or '**limits** for transparent use)
Specify lower left corner (page 753) or [ON (page 753)/OFF (page 753)] <*current*>: *Specify a point, enter* **on** *or* **off**, *or press* ENTER
**Lower-Left Corner** Specifies the lower-left corner for the grid limits.

Specify upper right corner <*current*>: *Specify a point or press* ENTER




**On** Turns on limits checking. When limits checking is on, you cannot enter points outside the grid limits. Because limits checking tests only points that you enter, portions of objects such as circles can extend outside the grid limits.

**Off** Turns off limits checking but maintains the current values for the next time you turn on limits checking.

# LINE

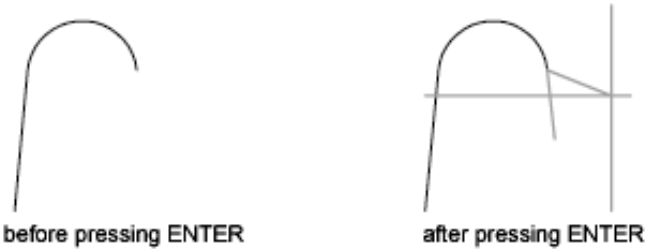
## Quick Reference

Creates straight line segments

Draw 
Draw ➤ LineAt the Command prompt, enter line.
**line**

Specify first point: *Specify a point or press* ENTER *to* continue (page 754) *from the last drawn line or arc*
Specify next point or [Close (page 754)/Undo (page 754)]:
**Continue** Continues a line from the endpoint of the most recently drawn line.


If the most recently drawn line is an arc, its endpoint defines the starting point of the line, and the line is drawn tangent to the arc.




**Close** Ends the last line segment at the beginning of the first line segment, which forms a closed loop of line segments. You can use Close after you have drawn a series of two or more segments.


**Undo** Erases the most recent segment of a line sequence.


Entering **u** more than once backtracks through line segments in the order you created them.

# LINETYPE

## Quick Reference

Loads, sets, and modifies linetypes

Format ➤ LinetypeAt the Command prompt, enter linetype.
**linetype** (or '**linetype** for transparent use)

The Linetype Manager (page 755) is displayed.

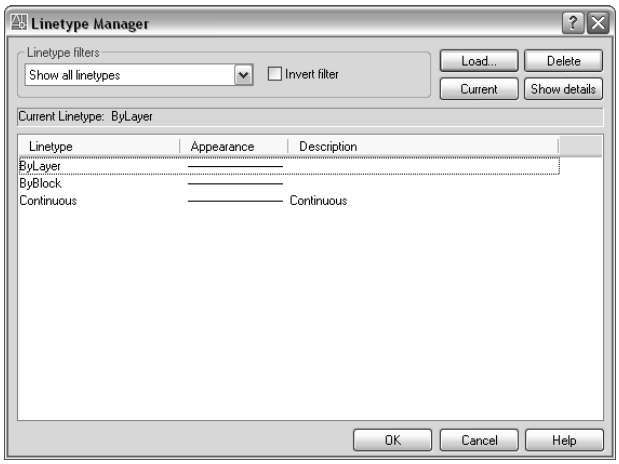
If you enter **-linetype** at the command prompt, options are displayed at the command prompt (page 759).

## Linetype Manager

### Quick Reference

Format ➤ LinetypeAt the Command prompt, enter linetype.
**linetype** (or '**linetype** for transparent use)

Loads linetypes and sets the current linetype.




### Linetype Filters

Determines which linetypes to display in the linetype list. You can filter linetypes based on whether they are xref-dependent, or whether they are referenced by objects.

**Invert Filter** Displays linetypes based on the opposites of the criteria you select. Linetypes that fit the inverse filter criteria are displayed in the linetype list.

### Load

Displays the Load or Reload Linetypes dialog box (page 758), in which you can load into the drawing selected linetypes from the *acad.lin* file and add them to the linetype list.

### Current

Sets the selected linetype to be the current linetype. Setting the current linetype to BYLAYER means that an object assumes the linetype that is assigned to a particular layer. Setting the linetype to BYBLOCK means that an object assumes the CONTINUOUS linetype until it is grouped into a block. Whenever the block is inserted, all objects inherit the block's linetype. The *CELTYPE* system variable stores the linetype name.

### Delete

Deletes selected linetypes from the drawing. You can delete only unused linetypes. The BYLAYER, BYBLOCK, and CONTINUOUS linetypes cannot be deleted.

**NOTE** Be careful about deleting linetypes if you are working on a drawing in a shared project or one based on a set of layering standards. The deleted linetype definition remains stored in the *acad.lin* or *acadiso.lin* file and can be reloaded.

### Show Details or Hide Details

Controls whether the Details section of the Linetype Manager is displayed.

### Current Linetype

Displays the current linetype name.

### List of Linetypes

Displays the loaded linetypes according to the option specified in Linetype Filters. To quickly select all or clear all linetypes, right-click in the linetype list to display the shortcut menu.

**Linetype** Displays names of loaded linetypes. To rename a linetype, select it and then click it again and enter a new name. BYLAYER, BYBLOCK, CONTINUOUS, and xref-dependent linetypes cannot be renamed. The linetype name can include up to 255 characters. Linetype names can contain letters, digits, blank spaces, and the special characters dollar sign ($), hyphen (-), and underscore (_). Linetype names cannot include the special characters comma (,), colon (:), equal sign (=), question mark (?), asterisk (*), less-than and greater-than symbols (> <), forward and back slashes (/ \), vertical bar (|), quote ("), or back quote (`).

**Appearance** Displays a sample of selected linetypes.

**Description** Displays descriptions of the linetypes, which can be edited in the Details area.

### Details

Provides alternative access to properties and additional settings.

**Name** Displays the selected linetype name, which can be edited. The linetype name can include up to 255 characters. Linetype names can contain letters,

digits, blank spaces, and the special characters dollar sign ($), hyphen (-), and underscore (_). Linetype names cannot include the special characters comma (,), colon (:), equal sign (=), question mark (?), asterisk (*), less-than and greater-than symbols (> <), forward and back slashes (/ \), vertical bar (|), quote ("), or back quote (`).

**Description** Displays the description of the selected linetype, which can be edited.

**Use Paper Space Units for Scaling** Scales linetypes in paper space and model space identically. Useful when working with multiple viewports. (*PSLTSCALE* system variable)

**Global Scale Factor** Displays the global scale factor for all linetypes. (*LTSCALE* system variable)

**Current Object Scale** Sets linetype scale for newly created objects. The resulting scale is the global scale factor multiplied by the object's scale factor. (*CELTSCALE* system variable)

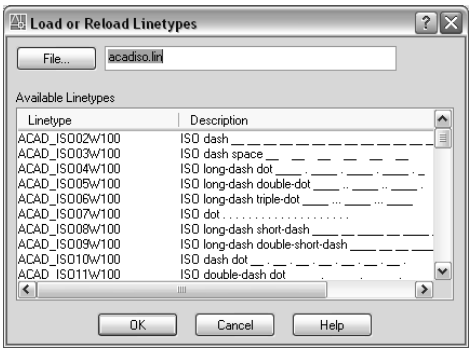
**ISO Pen Width** Sets the linetype scale to one of a list of standard ISO values. The resulting scale is the global scale factor multiplied by the object's scale factor.

# Load or Reload Linetypes Dialog Box

## Quick Reference

Format ➤ LinetypeAt the Command prompt, enter linetype.
**linetype** (or '**linetype** for transparent use)

Loads a linetype whose definition exists in a linetype library (LIN) file. The *acad.lin* file contains the standard linetypes.




**File Button** Displays the Select Linetype File dialog box, in which you can select a different linetype (LIN) file.

**File Name** Displays the name of the current LIN file. You can enter the name of another LIN file or click the File button to select a file from the Select Linetype File dialog box.

**Available Linetypes** Displays the linetypes available to load. To select or clear all of the linetypes on the list, right-click and choose Select All or Clear All.

# -LINETYPE

## Quick Reference

If you enter **-linetype** at the command prompt, the following LINETYPE command prompts are displayed.

Enter an option [? (page 759)/Create (page 759)/Load (page 760)/Set (page 760)]:
**?—List Linetypes** Displays the Select Linetype File dialog box (a standard file selection dialog box (page 931)). After you select an LIN file, the linetypes available in the file are listed.

**Create** Creates a new linetype and stores it in an LIN file.

Enter name of linetype to create:

The Create or Append Linetype File dialog box (a standard file selection dialog box (page 931)) is displayed. Specify the file to which you want the linetype added.

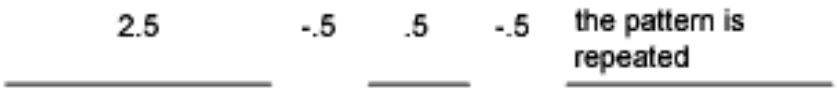
Descriptive text: *Enter optional descriptive text*

Enter a linetype description up to 47 characters long. The description can be a comment or a series of underscores, dots, dashes, and spaces to show a simple representation of the linetype pattern.

Enter linetype pattern (on next line):

A,

Enter a pattern definition as a series of numbers separated by commas. Enter positive values to specify lengths of dashes, and enter negative values to specify lengths of spaces. Use a zero to represent a dot.




The "A" in the pattern definition prompt specifies the pattern alignment used at the ends of individual lines, circles, and arcs. Only A-type alignment is supported. With A-type alignment, lines and arcs are guaranteed to start and end with a dash. The A is automatically included in the definition. If you use a text editor to create a linetype, you must enter **a** at the beginning of the definition.

After creating a linetype, you must load it to make it accessible.

You cannot create complex linetypes with LINETYPE. For more information, see "Custom Linetypes" in the *Customization Guide*.

**Load** Loads a linetype whose definition exists in a file. The *acad.lin* file contains the standard linetypes.

Enter linetype(s) to load: *Enter a name or a list of names separated by commas*

The Select Linetype File dialog box (a Standard File Selection Dialog Boxes (page 931)standard file selection dialog box) is displayed. Enter or select the file in which the linetype you want to load is stored.

**Set** Sets the current linetype for objects that will be drawn subsequently. You can control the linetype of objects individually or by layer.

Specifies linetype name or [?] <*current*>: *Enter a linetype name,* **?** *to list all loaded linetype names,* **bylayer***, or* **byblock***, or press* ENTER

The linetype you enter becomes the current linetype. All new objects are drawn with this linetype, regardless of the current layer. If the linetype you request is not loaded, the program searches for its definition in the *acad.lin* file. If the linetype is neither loaded nor in *acad.lin*, the program displays a message and returns you to the Command prompt.

Enter **?** to list all loaded linetype names. If you enter **bylayer**, new objects inherit the linetype associated with the layer on which the object is drawn.

If you enter **byblock**, new objects are drawn using the CONTINUOUS linetype until they are grouped into a block. Whenever you insert that block, the objects inherit the linetype of the block.

# LIST

## Quick Reference

Displays database information for selected objects

Inquiry 
Tools ➤ Inquiry ➤ ListAt the Command prompt, enter list.
**list**
Select objects: *Use an object selection method*

The text window displays the object type, object layer, and *X*,*Y*,*Z* position relative to the current user coordinate system (UCS) and whether the object is in model space or paper space.

LIST reports color, linetype, and lineweight information if these items are not set to BYLAYER. The thickness of an object is displayed if it is nonzero. *Z* coordinate information defines the elevation. If the extrusion direction of the entry differs from the *Z* axis (0,0,1) of the current UCS, LIST also reports the extrusion direction in UCS coordinates.

LIST reports additional information related to the specific object selected.

# LIVESECTION

## Quick Reference

Turns on live sectioning for a selected section object.

livesection
Select section object: *Select a section object*

Live sectioning is turned on for the selected section object. When turned on, the interior of 3D objects interesected by the section object are displayed.

# LOAD

## Quick Reference

Makes shapes available for use by the SHAPE command

**load**

The Select Shape File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the shape file name, or select a file name from the list.

You must load a shape (SHP) file the first time you need it; it is loaded automatically thereafter. The shape file must be available each time you edit the drawing.

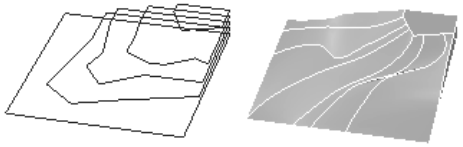
# LOFT

## Quick Reference

Creates a 3D solid or surface by lofting through a set of two or more curves

Modeling 
Draw ➤ Modeling ➤ LoftAt the Command prompt, enter loft.
**loft**
3D Make panel, Loft

With the LOFT command, you can create a new solid or surface by specifying a series of cross sections. The cross sections define the profile (shape) of the resulting solid or surface. Cross sections (generally, curves or lines) can be open (for example, an arc) or closed (for example, a circle). LOFT draws a solid or surface in the space between the cross sections. You must specify at least two cross sections when you use the LOFT command.



Cross sections Lofted solid

You can use the following objects when creating a lofted solid or surface:

| Objects That Can Be Used as Cross Sections | Objects That Can Be Used as a Loft Path | Objects That Can Be Used as Guides |
|---|---|---|
| Line | Line | Line |
| Arc | Arc | Arc |
| Elliptical arc | Elliptical arc | Elliptical arc |
| 2D polyline | Spline | 2D spline |
| 2D spline | Helix | 3D spline |
| Circle | Circle | 2D polyline |
| Ellipse | Ellipse | 3D polyline |
| Points (first and last cross section only) | 2D polyline | |
| Region | 3D polyline | |
| Planar face of solid | | |
| Planar surface | | |
| Planar 3D face | | |
| 2D solid | | |
| Trace | | |

With the Path option, you can select a single path curve to define the shape of the solid or surface. With the Guides option, you can select multiple curves to define the contours of the solid or surface.

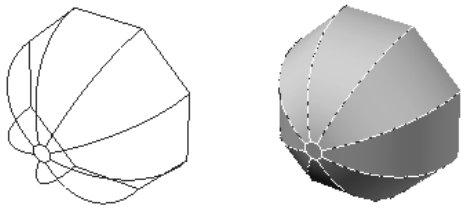
The *DELOBJ* system variable controls whether the cross sections, guides, and path are automatically deleted when the solid or surface is created or whether you are prompted to delete these objects.

You can select the cross sections before you start the command.

Select cross sections in lofting order: *Select open or closed curves in the order that the surface or solid will pass through*
Enter an option [Guides (page 764)/Path (page 765)/Cross sections only (page 765)] <Cross sections only>: *Press* ENTER *to use the selected cross sections, displaying the Loft Settings dialog box, or enter an option*

### Guides

Specifies guide curves that control the shape of the lofted solid or surface. Guide curves are lines or curves that further define the form of the solid or surface by adding additional wireframe information to the object. You can use guide curves to control how points are matched up on corresponding cross sections to prevent undesired results, such as wrinkles in the resulting solid or surface.



Cross sections with guide curves

Lofted solid

Each guide curve must meet the following criteria to work correctly:

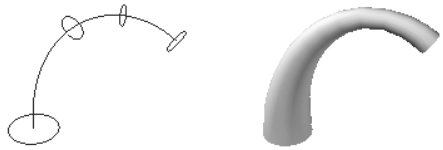
- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

You can select any number of guide curves for the lofted surface or solid.

Select guide curves: *Select the guide curves for the lofted solid or surface, and then press* ENTER

#### Path

Specifies a single path for the lofted solid or surface.



Cross sections with path curve

Lofted solid

The path curve must intersect all planes of the cross sections.

Select path: *Specify a single path for the lofted solid or surface*

#### Cross Sections Only

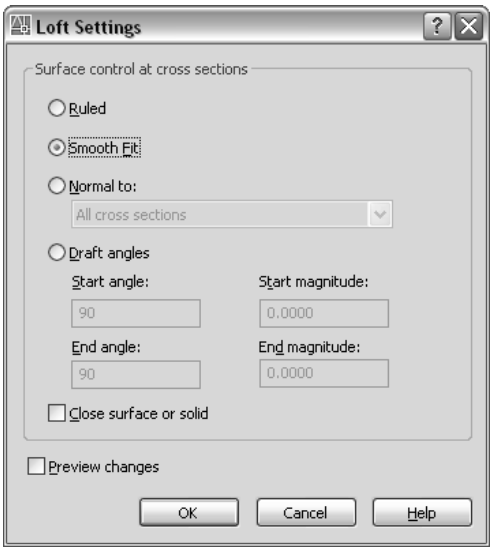
Displays the Loft Settings dialog box.

## Loft Settings Dialog Box

### Quick Reference

Modeling
Draw ➤ Modeling ➤ LoftAt the Command prompt, enter loft.
**loft**

Controls the contour of a lofted surface at its cross sections. Also allows you to close the surface or solid.




### Ruled

Specifies that the solid or surface is ruled (straight) between the cross sections and has sharp edges at the cross sections. (*LOFTNORMALS* system variable)

### Smooth Fit

Specifies that a smooth solid or surface is drawn between the cross sections and has sharp edges at the start and end cross sections. (*LOFTNORMALS* system variable)

### Normal to

Controls the surface normal of the solid or surface where it passes through the cross sections. (*LOFTNORMALS* system variable)

**Start Cross Section** Specifies that the surface normal is normal to the start cross section.

**End Cross Section** Specifies that the surface normal is normal to the end cross section.

**Start and End Cross Sections** Specifies that the surface normal is normal to both the start and end cross sections.

**All Cross Sections** Specifies that the surface normal is normal to all cross sections.

### Draft Angles

Controls the draft angle and magnitude of the first and last cross sections of the lofted solid or surface. The draft angle is the beginning direction of the surface. 0 is defined as outward from the plane of the curve. (*LOFTNORMALS* system variable)


**Start Angle** Specifies the draft angle for the start cross section. (*LOFTANG1* system variable)

**Start Magnitude** Controls the relative distance of the surface from the start cross section in the direction of the draft angle before the surface starts to bend toward the next cross section. (*LOFTMAG1* system variable)

**End Angle** Specifies the draft angle for the end cross section. (*LOFTANG2* system variable)

**End Magnitude** Controls the relative distance of the surface from the end cross section in the direction of the draft angle before the surface starts to bend toward the previous cross section. (*LOFTMAG2* system variable)

### Close Surface or Solid

Closes and opens a surface or solid. When using this option, the cross sections should form a torus-shaped pattern so that the lofted surface or solid can form a closed tube. (*LOFTPARAM* system variable)

### Preview Changes

Applies the current settings to the lofted solid or surface and displays a preview in the drawing area.

# LOGFILEOFF

## Quick Reference

Closes the text window log file opened by LOGFILEON

**logfileoff**

The program stops recording the text window contents and closes the log file. You can also control the log file with the OPTIONS (page 946) command. Use the Maintain a Log File option on the Open and Save tab (page 957) in the Options dialog box to turn the log file off and on. Use the Files tab (page 948) to change the location of the log file.

Each drawing saves a log file (with the extension *.log*) that may need periodic deletion as the number of log files continues to grow.

# LOGFILEON

## Quick Reference

Writes the text window contents to a file

**logfileon**

The contents of the text window are recorded in the log file until you exit the program or use the *LOGFILEOFF* command.

You can also control the log file with the OPTIONS (page 946) command. Use the Maintain a Log File option on the Open and Save tab (page 957) in the Options dialog box to turn the log file off and on. Use the Files tab (page 948) to change the location of the log file.

Each drawing saves a log file (with the extension *.log*) that may need periodic deletion as the number of log files continues to grow.

# LSEDIT

## Quick Reference

Has no effect except to preserve the integrity of scripts

**lsedit**

Obsolete command.

# LSLIB

## Quick Reference

Has no effect except to preserve the integrity of scripts

**lslib**

Obsolete command.

# LSNEW

## Quick Reference

Has no effect except to preserve the integrity of scripts

**lsnew**

Obsolete command.

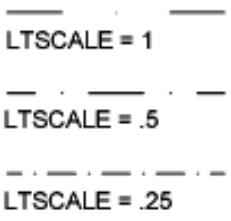
# LTSCALE

## Quick Reference

Sets the global linetype scale factor

**ltscale** (or '**ltscale** for transparent use)
Enter new linetype scale factor <*current*>: *Enter a positive real value or press* ENTER

Use LTSCALE to change the scale factor of linetypes for all objects in a drawing. Changing the linetype scale factor causes the drawing to be regenerated.




# LWEIGHT

## Quick Reference

Sets the current lineweight, lineweight display options, and lineweight units

Format ➤ LineweightAt the Command prompt, enter lweight.
Right-click LWT on the status bar and choose Settings.
**lweight** (or '**lweight** for transparent use)

The Lineweight Settings dialog box (page 770) is displayed.

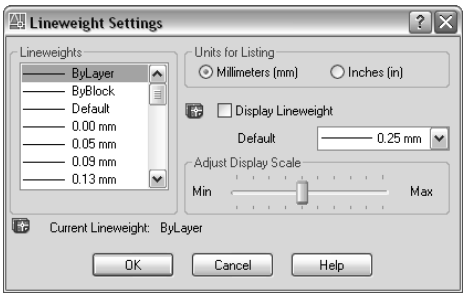
If you enter **-lweight** at the command prompt, options are displayed at the command prompt (page 772).

# Lineweight Settings Dialog Box

## Quick Reference

Format ➤ LineweightAt the Command prompt, enter lweight.
Right-click LWT on the status bar and choose Settings.
**lweight** (or '**lweight** for transparent use)

Sets the current lineweight, sets the lineweight units, controls the display and display scale of lineweights, and sets the DEFAULT lineweight value for layers. For a table of valid lineweights, see "Overview of Lineweights" in the *User's Guide*.




### Lineweights

Displays the available lineweight values. Lineweight values consist of standard settings including BYLAYER, BYBLOCK, and DEFAULT. The DEFAULT value is set by the *LWDEFAULT* system variable, which has an initial value of 0.01 inches or 0.25 mm. All new layers use the default setting. The lineweight value of 0 plots at the thinnest lineweight available on the specified plotting device and is displayed at one pixel wide in model space.

### Current Lineweight

Displays the current lineweight. To set the current lineweight, select a lineweight from the lineweight list and choose OK.

### Units for Listing

Specifies whether lineweights are displayed in millimeters or inches. You can also set Units for Listing by using the *LWUNITS* system variable.

**Millimeters (mm)** Specifies lineweight values in millimeters.

**Inches (in.)** Specifies lineweight values in inches.

### Display Lineweight

Controls whether lineweights are displayed in the current drawing. If this option is selected, lineweights are displayed in model space and paper space. You can also set Display Lineweight by using the *LWDISPLAY* system variable. Regeneration time increases with lineweights that are represented by more than one pixel. Clear Display Lineweight if performance slows down when working with lineweights turned on in a drawing. This option does not affect how objects are plotted.

#### Default

Controls the DEFAULT lineweight for layers. The initial DEFAULT lineweight is 0.01 inches or 0.25 mm. (LWDEFAULT system variable)

#### Adjust Display Scale

Controls the display scale of lineweights on the Model tab. On the Model tab, lineweights are displayed in pixels. Lineweights are displayed using a pixel width in proportion to the real-world unit value at which they plot. If you are using a high-resolution monitor, you can adjust the lineweight display scale to better display different lineweight widths. The Lineweight list reflects the current display scale.

Objects with lineweights that are displayed with a width of more than one pixel may increase regeneration time. If you want to optimize performance when working in the Model tab, set the lineweight display scale to the minimum value or turn off lineweight display altogether.

## -LWEIGHT

### Quick Reference

If you enter **-lweight** at the command prompt, the following LWEIGHT command prompts are displayed.

Current lineweight: *current*
Enter default lineweight (page 772) for new objects or [? (page 773)]: *Enter a valid lineweight or enter* **?**

The current lineweight value is displayed; if the value is not BYLAYER, BYBLOCK or DEFAULT, the value is displayed in millimeters or inches.

**Default Lineweight** Sets the current default lineweight. Lineweight values consist of fixed settings, including BYLAYER, BYBLOCK, and DEFAULT. Values are calculated in either inches or millimeters; millimeters are the default. If you enter a valid lineweight value, the current default lineweight is set to the new value. If you enter any other value, the default is set to the nearest valid value.

To plot an object with a lineweight that is not found in the list of fixed lineweight values, you can use the Plot Style Table Editor to customize plotted lineweights. See Control Plotted Lineweight and Linetype in the *User's Guide*. The DEFAULT value is set by the LWDEFAULT system variable and has an

initial value of 0.01 inches or 0.25 mm. The lineweight value of 0 plots at the thinnest lineweight available on the specified plotting device and is displayed at a value of one pixel in model space.

**?—List Lineweights** Displays a list of valid lineweight values in the current lineweight units.

---

**NOTE** If you save a drawing using the AutoCAD Release 14, or earlier, format, the drawing preview displays lineweights even though the drawing saved in the earlier format does not display lineweights.

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# M Commands

# 13

**In this chapter**

- MLEADERSTYLE
- MLEDIT
- MLINE
- MLSTYLE
- MODEL
- MOVE
- MREDO
- MSLIDE
- MSPACE
- MTEDIT
- MTEXT
- MULTIPLE
- MVIEW
- MVSETUP

# MARKUP

## Quick Reference

Displays the details of markups and allows you to change their status

Standard 
Tools ➤ Palettes ➤ Markup Set ManagerAt the Command prompt, enter markup.
**markup**

Opens the Markup Set Manager (page 777).

# Markup Set Manager

## Quick Reference

Standard
Tools ➤ Palettes ➤ Markup Set ManagerAt the Command prompt, enter markup.
**markup**

Displays information about loaded markup sets and their status. You can show or hide markups and their originating drawing files in your drawing area. You can change the status of individual markups and add notes and comments to them.

### Markup Set List Control

The Markup Set List control displays the name of the markup set, or, if no markup sets are open, the Open option. The Markup Set List control provides the following options.

**Names of Open Markup Sets** Lists all open markup sets, if any. A check is displayed next to the current markup set. The current markup set is the open markup set that is displayed in the Markup Set Manager window.

**Recent** Displays a list of recently opened markup sets.

**Open** Displays the Open Markup DWF dialog box (a standard file selection dialog box), in which you can select a Design Web Format™ (DWF™ ) file that contains markups. When you click Open, the markups in the selected DWF file are loaded into the Markup Set Manager.

### Republish Markup DWF

Provides options for republishing the marked-up DWF file. If any sheets were added to the DWF file in Autodesk® Design Review, those sheets will not be included in the republished DWF file.

**Republish All Sheets** Republishes all sheets in the marked-up DWF file. Opens the Select DWF File dialog box (a standard file selection dialog box), in which you can select the previously published DWF to overwrite it, or you can enter a new name for the DWF file. When you click Select, the previously published DWF file is overwritten or a new DWF file is created that contains any changes you made to the drawing file geometry and the status of the markups.

**Republish Markup Sheets** Republishes only those sheets in the marked-up DWF file that have associated markups. Opens the Select DWF File dialog box (a standard file selection dialog box), in which you can select the previously published DWF to overwrite it, or you can enter a new name for the DWF file. When you click Select, the previously published DWF file is overwritten or a new DWF file is created that contains any changes you made to the drawing file geometry and the status of the markups.

### View Redline Geometry

Displays or hides redline markup geometry in the drawing area. When this button is displayed as selected, redline markup geometry is displayed in the drawing area.

### View DWG Geometry

Displays or hides the originating drawing file in the drawing area. When this button is displayed as selected, the drawing file is displayed in the drawing area.

### View DWF Geometry

Displays or hides the DWF file geometry in the drawing area. When this button is displayed as selected, the DWF file geometry is displayed in the drawing area.

### Markups

Displays loaded markup sets. The top-level node in the tree view represents the currently loaded markup set. A drawing sheet node is displayed for each drawing sheet that has associated markups. Any sheets that were added to the DWF file in Autodesk Design Review are listed in italics. Each markup is displayed as an individual node under its corresponding drawing sheet. The icon associated with each markup represents the status of the markup. You can double-click any of the markups in the tree view to view the associated drawing sheet in the drawing area. When you double-click either a sheet that was added in Autodesk Design Review or a markup for that sheet, that sheet in the DWF file will open in Autodesk DWF Viewer (or in Autodesk Design Review, if you have it installed).

**Markup Set Node** Indicates the currently loaded markup set.

**Drawing Sheet Node** Indicates a drawing sheet that has one or more associated markups. Only drawing sheets that have corresponding markups are displayed in the tree view of the Markups area. You can double-click a drawing sheet node to open the originating drawing file for that drawing sheet.

**<None>** Indicates an individual markup that does not yet have an assigned status. This is the default status for new markups.

**Question** Indicates an individual markup that has an assigned status of Question. When you open and view a markup, you can change the status to Question if you need additional information about the markup.

**For Review** Indicates an individual markup that has an assigned status of For Review. When you implement a markup, you can change the status to For Review to indicate that the markup creator should review the changes to the drawing sheet and the status of the markup.

**Done** Indicates an individual markup that has an assigned status of Done. When a markup has been implemented and reviewed, you can change the status to Done.

### Markup Shortcut Menu Options (Markup Set Node)

Right-clicking a markup set node in the Markups area displays the following shortcut menu options.

**Open Markup DWF** Opens the Open Markup DWF dialog box (a standard file selection dialog box), in which you can select a DWF file that contains markups. When you click Open, the markups in the selected DWF file are loaded into the Markup Set Manager.

**Close Markup DWF** Closes the selected marked-up DWF and removes it from the Markup Set Manager.

**Save Markup History Changes** Saves changes to the status of markups and added comments to the marked-up DWF file.

### Markup Shortcut Menu Options (Drawing Sheet Node)

Right-clicking a drawing sheet node in the Markups area displays the following shortcut menu options.

**Open Sheet** Opens the originating drawing file for that drawing sheet and makes that layout the active layout in the drawing area.

**Republish Markup DWF** Opens the Select DWF File dialog box (a standard file selection dialog box), in which you can select the previously published DWF to overwrite or enter a new name for the DWF file. When you click Select, a new DWF file is created that contains any changes you made to the drawing file geometry and the status of the markups.

### Markup Shortcut Menu Options (Markup Node)

Right-clicking an individual markup node in the Markups area displays the following shortcut menu options.

**Open Markup** Opens the originating drawing file for the drawing sheet associated with the selected markup and makes that layout the active layout in the drawing area. The associated markup DWF file is also opened as a read-only reference in the drawing area.

**Markup Status** Displays a shortcut menu that provides a list of status options. A check mark precedes the currently assigned status for the selected markup.

- *<None>:* Indicates that the markup does not yet have an assigned status. This is the default status for new markups.
- *Question:* After you open and view a markup, you can change its status to Question if you need additional information about the markup.

- *For Review:* After you have implemented a markup, you can change the status to For Review to indicate that the markup creator should review the changes to the drawing sheet and the status of the markup.
- *Done:* After a markup has been implemented and reviewed, you can change its status to Done.

**Restore Initial Markup View** If you panned or zoomed the selected markup, or rotated the view in model space, restores the original view of the selected markup.

**Republish Markup DWF** Opens the Select DWF File dialog box (a standard file selection dialog box), in which you can select the previously published DWF to overwrite or enter a new name for the DWF file. When you click Select, the previously published DWF file is overwritten or a new DWF file is created that contains any changes you made to the drawing file geometry and the status of the markups.

### Show All Sheets/Hide Non-Markup Sheets

Displays all sheets in the marked-up DWF file in the tree view of the Markup Set Manager, or hides the sheets that do not have associated markups.

### Markup Details

Provides information about the currently selected node (markup set, drawing sheet, or individual markup) in the Markups area.

When an individual markup is selected in the Markups area, Markup Details displays the status of the markup. You can change the status.

The Markup History area (in the lower portion of the Markup Details area) provides a noneditable history log of the selected markup in reverse chronological order. In the Notes area, just below the Markup History area, you can add notes and comments that are automatically saved with the markup.

**Preview** Changes the Markup Details area to the Markup Preview area and displays a thumbnail image of the selected drawing sheet or markup. You can click the Details button to change the area back to the Markup Details area.

**Details** Changes the Markup Preview area to the Markup Details area, which provides information about the selected node in the Markups area. You can click the Preview button to change the area back to the Markup Preview area.

**Markup Preview**

Displays a thumbnail preview of the currently selected drawing sheet or markup.

**Details** Changes the Markup Preview area to the Markup Details area, which provides information about the selected node in the Markups area. You can click the Preview button to change the area back to the Markup Preview area.

**Preview** Changes the Markup Details area to the Markup Preview area and displays a thumbnail image of the selected drawing sheet or markup. You can click the Details button to change the area back to the Markup Details area.

# MARKUPCLOSE

## Quick Reference

Closes the Markup Set Manager

Standard 

Tools ➤ Palettes ➤ Markup Set ManagerAt the Command prompt, enter markupclose.

**markupclose**

The Markup Set Manager (page 777) window closes.

# MASSPROP

## Quick Reference

Calculates the mass properties of regions or 3D solids

Inquiry 

Tools ➤ Inquiry ➤ Region/Mass PropertiesAt the Command prompt, enter massprop.

**massprop**

Select objects: *Use an object selection method*

If you select multiple regions, only those that are coplanar with the first selected region are accepted.

MASSPROP displays the mass properties in the text window, and then asks if you want to write the mass properties to a text file.

Write analysis to a file? <N>: *Enter* **y** *or* **n***, or press* ENTER

If you enter **y**, MASSPROP prompts you to enter a file name. The default extension for the file is *.mpr*, but it is a text file that can be opened with any text editor.

The properties that MASSPROP displays depend on whether the selected objects are regions (page 783), and whether the selected regions are coplanar with the *XY* plane of the current user coordinate system (UCS), or solids (page 785). For a list of the parameters that control the MASSPROP units, see Calculations Based on the Current UCS (page 786).

### Regions

The following table shows the mass properties that are displayed for all regions.

**Mass properties for all regions**

| Mass property | Description |
|---|---|
| Area | The surface area of solids or the enclosed area of regions. |

**Mass properties for all regions**

| Mass property | Description |
|---|---|
| Perimeter | The total length of the inside and outside loops of a region. The perimeter of a solid is not calculated. |
| Bounding box | The two coordinates that define the bounding box. For regions that are coplanar with the *XY* plane of the current user coordinate system, the bounding box is defined by the diagonally opposite corners of a rectangle that encloses the region. For regions that are not coplanar with the *XY* plane of the current UCS, the bounding box is defined by the diagonally opposite corners of a 3D box that encloses the region. |
| Centroid | A 2D or 3D coordinate that is the center of area for regions. For regions that are coplanar with the *XY* plane of the current UCS, this coordinate is a 2D point. For regions that are not coplanar with the *XY* plane of the current UCS, this coordinate is a 3D point. |

If the regions are coplanar with the *XY* plane of the current UCS, the additional properties shown in the following table are displayed.

**Additional mass properties for coplanar regions**

| Mass property | Description |
|---|---|
| Moments of inertia | A value used when computing the distributed loads, such as fluid pressure on a plate, or when calculating the forces inside a bending or twisting beam. The formula for determining area moments of inertia is<br>area_moments_of_inertia = area_of_interest * radius2<br>The area moments of inertia has units of distance to the fourth power. |
| Products of inertia | Property used to determine the forces causing the motion of an object. It is always calculated with respect to two orthogonal planes. The formula for product of inertia for the *YZ* plane and *XZ* plane is<br>product_of_inertiaYZ,XZ = mass * distcentroid_to_YZ * distcentroid_to_XZ<br>This *XY* value is expressed in mass units times the length squared. |

**Additional mass properties for coplanar regions**

| Mass property | Description |
|---|---|
| Radii of gyration | Another way of indicating the moments of inertia of a solid. The formula for the radii of gyration is gyration_radii = (moments_of_ inertia/body_mass)1/2 Radii of gyration are expressed in distance units. |
| Principal moments and *X,Y,Z* directions about centroid | Calculations that are derived from the products of inertia and that have the same unit values. The moment of inertia is highest through a certain axis at the centroid of an object. The moment of inertia is lowest through the second axis that is normal to the first axis and that also passes through the centroid. A third value included in the results is somewhere between the high and low values. |

### Solids

The following table shows the mass properties that are displayed for solids.

**Mass properties for solids**

| Mass property | Description |
|---|---|
| Mass | The measure of inertia of a body. Because a density of one is used, mass and volume have the same value. |
| Volume | The amount of 3D space that a solid encloses. |
| Bounding box | The diagonally opposite corners of a 3D box that encloses the solid. |
| Centroid | A 3D point that is the center of mass for solids. A solid of uniform density is assumed. |
| Moments of inertia | The mass moments of inertia, which is used when computing the force required to rotate an object about a given axis, such as a wheel rotating about an axle. The formula for mass moments of inertia is mass_moments_of_inertia = object_mass * radiusaxis2 |

**Mass properties for solids**

| Mass property | Description |
|---|---|
| | Mass moments of inertia unit is mass (grams or slugs) times the distance squared. |
| Products of inertia | Property used to determine the forces causing the motion of an object. It is always calculated with respect to two orthogonal planes. The formula for product of inertia for the *YZ* plane and *XZ* plane is product_of_inertiaYZ,XZ = mass * distcentroid_to_YZ * distcentroid_to_XZ<br>This *XY* value is expressed in mass units times the length squared. |
| Radii of gyration | Another way of indicating the moments of inertia of a solid. The formula for the radii of gyration is gyration_radii = (moments_of_inertia/body_mass)1/2<br>Radii of gyration are expressed in distance units. |
| Principal moments and *X,Y,Z* directions about centroid | Calculations that are derived from the products of inertia and that have the same unit values. The moment of inertia is highest through a certain axis at the centroid of an object. The moment of inertia is lowest through the second axis that is normal to the first axis and that also passes through the centroid. A third value included in the results is somewhere between the high and low values. |

### Calculations Based on the Current UCS

The following table shows the parameters that control the units in which mass properties are calculated.

**Parameters that control MASSPROP units**

| Parameter | Used to calculate |
|---|---|
| DENSITY | Mass of solids |
| LENGTH | Volume of solids |

**Parameters that control MASSPROP units**

| Parameter | Used to calculate |
|---|---|
| LENGTH*LENGTH | Area of regions and surface area of solids |
| LENGTH*LENGTH*LENGTH | Bounding box, radii of gyration, centroid, and perimeter |
| DENSITY*LENGTH*LENGTH | Moments of inertia, products of inertia, and principal moments |

# MATCHCELL

## Quick Reference

Applies the properties of a selected table cell to other table cells

With a table selected and a cell selected, right-click and click Match Cell.
**matchcell**
Select source cell: *Click inside a cell in a table whose properties you want to copy*
Select destination cell: *Click inside table cells to copy properties from the source cell, and right-click or press* ENTER *or* ESC *to exit*

All the properties of the source cell are copied to the destination cells except for the cell type: text or block.

# MATCHPROP

## Quick Reference

Applies the properties of a selected object to other objects

Standard 
Modify ➤ Match PropertiesAt the Command prompt, enter matchprop.

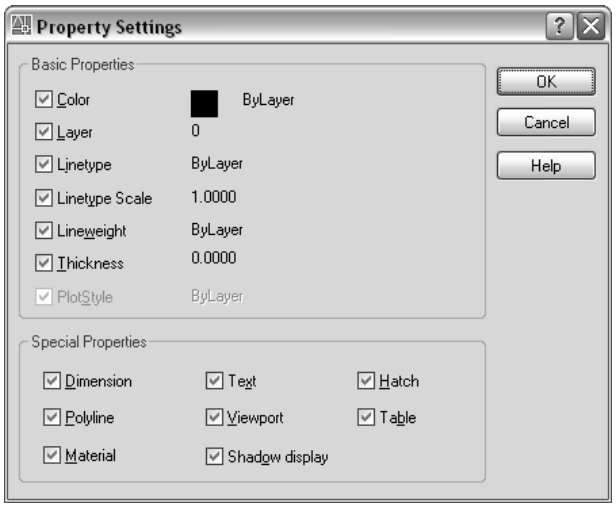
**matchprop** or **painter** (or '**matchprop** for transparent use)
Select source object: *Select the object whose properties you want to copy*
Current active settings: *Currently selected matchprop settings*
Select destination object(s) or [Settings]: *Enter* **s** *or select one or more objects to copy properties to*
**Destination Object(s)** Specifies the objects to which you want to copy the properties of the source object. You can continue selecting destination objects, or press ENTER to apply the properties and end the command.

**Settings** Displays the Property Settings dialog box (page 788), in which you can control which object properties to copy to the destination objects. By default, in the Property Settings dialog box all object properties are selected for copying.

# Property Settings Dialog Box

## Quick Reference

Standard
Modify ➤ Match PropertiesAt the Command prompt, enter matchprop.
**matchprop** or **painter** (or '**matchprop** for transparent use)




Specifies which basic properties and special properties to copy from the source object to the destination objects.

**Color** Changes the color of the destination object to that of the source object. Available for all objects.

**Layer** Changes the layer of the destination object to that of the source object. Available for all objects.

**Linetype** Changes the linetype of the destination object to that of the source object. Available for all objects except attributes, hatches, multiline text, points, and viewports.

**Linetype Scale** Changes the linetype scale factor of the destination object to that of the source object. Available for all objects except attributes, hatches, multiline text, points, and viewports.

**Lineweight** Changes the lineweight of the destination object to that of the source object. Available for all objects.

**Thickness** Changes the thickness of the destination object to that of the source object. Available only for arcs, attributes, circles, lines, points, 2D polylines, regions, text, and traces.

**Plot Style** Changes the plot style of the destination object to that of the source object. If you are working in color-dependent plot style mode (*PSTYLEPOLICY* is set to 1), this option is unavailable. Available for all objects, except those with the Jitter edge modifier applied.

**Dimension** In addition to basic object properties, changes the dimension style and properties of the destination object to that of the source object. Available only for dimension, leader, and tolerance objects.

**Polyline** In addition to basic object properties, changes the width and linetype generation properties of the destination polyline to those of the source polyline. The fit/smooth property and the elevation of the source polyline are not transferred to the destination polyline. If the source polyline has variable width, the width property is not transferred to the destination polyline.

**Material** In addition to basic object properties, changes the material applied to the object. If the source object does not have a material assigned and the destination object does, the material is removed from the destination object.

**Text** In addition to basic object properties, changes the text style and properties of the destination object to that of the source object. Available only for single-line and multiline text objects.

**Viewport** In addition to basic object properties, changes the following properties of the destination paper space viewport to match those of the source viewport: on/off, display locking, standard or custom scale, shade plot, snap, grid, and UCS icon visibility and location.

The settings for clipping and for UCS per viewport and the freeze/thaw state of the layers are not transferred to the destination object.

**Shadow Display** In addition to basic object properties, changes the shadow display. The object can cast shadows, receive shadows, or both, or it can ignore shadows.

**Hatch** In addition to basic object properties, changes the hatch properties (including its properties) of the destination object to that of the source object. To match the hatch origin, use Inherit Properties in HATCH or HATCHEDIT. Available only for hatch objects.

**Table** In addition to basic object properties, changes the table style of the destination object to that of the source object. Available only for table objects.

**Multileader** In addition to basic object properties, changes the multileader style and properties of the destination object to that of the source object. Available only for multileader objects.

# MATERIALATTACH

## Quick Reference

Applies materials to objects by layer

**materialattach**
Materials panel, Attach by Layer

The Material Attachment Options dialog box (page 791) is displayed.

# Material Attachment Options Dialog Box

## Quick Reference


**materialattach**

Associates materials with layers. All objects on the layer that have their Material property set to BYLAYER have the specified material applied.

**Materials List** Lists all the materials in the drawing, both in use and not in use. Drag a material onto a layer in the Layers List to apply the material to that layer.

**Layers List** Lists all layers in the drawing and in any external references applied to the drawing. When a material is applied to the layer, that material is displayed next to the layer. Use the Detach button to remove the material from the layer.

---

**NOTE** If you change the association of a material to a layer in an xref, the change is not saved back to the xref drawing.

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# MATERIALMAP

## Quick Reference

Displays a material mapping gizmo to adjust the mapping on a face or an object

Mapping
Render
**materialmap**
Materials panel, Mapping flyout (Box Mapping, Planar Mapping, Spherical Mapping, Cylindrical Mapping)
Select an option [Box (page 792)/Planar (page 792)/Spherical (page 793)/Cylindrical (page 793)/copY mapping to (page 793)/Reset mapping (page 793)] <*current*>:

### Box

Select faces or objects:

The selection set must include at least one of the following object types: 3D solid, 3D surface, face, or 2D object with thickness.

Accept the mapping or [Move/Rotate/reseT/sWitch mapping mode]:
**Move** Displays the Move grip tool to move the map.

**Rotate** Displays the Rotate grip tool to rotate the map.

**Reset** Resets the UV coordinates to the default for the map.

**Switch Mapping Mode** Re-displays the main command prompt of options.[Box (page 792)/Planar (page 792)/Spherical (page 793)/Cylindrical (page 793)/copY mapping to (page 793)/Reset mapping (page 793)] <*current*>:

### Planar

Accept the mapping or [Move/Rotate/reseT/sWitch mapping mode]:
**Move** Displays the Move grip tool to move the map.

**Rotate** Displays the Rotate grip tool to rotate the map.

**Reset** Resets the UV coordinates to the default for the map.

**Switch Mapping Mode** Re-displays the main command prompt. [Box (page 792)/Planar (page 792)/Spherical (page 793)/Cylindrical (page 793)/copY mapping to (page 793)/Reset mapping (page 793)] <*current*>:

### Cylindrical

Accept the mapping or [Move/Rotate/reseT/sWitch mapping mode]:
**Move** Displays the Move grip tool to move the map.

**Rotate** Displays the Rotate grip tool to rotate the map.

**Reset** Resets the UV coordinates to the default for the map.

**Switch Mapping Mode** Re-displays the main command prompt. [Box (page 792)/Planar (page 792)/Spherical (page 793)/Cylindrical (page 793)/copY mapping to (page 793)/Reset mapping (page 793)] <*current*>:

### Spherical

Accept the mapping or [Move/Rotate/reseT/sWitch mapping mode]:
**Move** Displays the Move grip tool to move the map.

**Rotate** Displays the Rotate grip tool to rotate the map.

**Reset** Resets the UV coordinates to the default for the map.

**Switch Mapping Mode** Re-displays the main command prompt. [Box (page 792)/Planar (page 792)/Spherical (page 793)/Cylindrical (page 793)/copY mapping to (page 793)/Reset mapping (page 793)] <*current*>:

### Copy Mapping To

Select the objects to copy the mapping to:

Applies mapping from the original object or face to the selected objects.

### Reset Mapping

Resets the UV coordinates to the default for the map.

# MATERIALS

## Quick Reference

Manages, applies, and modifies materials

Render 
View ➤ Render ➤ MaterialsAt the Command prompt, enter materials.
**materials**
Materials panel, Materials

The Materials window (page 794) is displayed.

---

**NOTE** When Texture Compression is turned on, the amount of video memory required to open a drawing that contains materials with images is decreased. By using the effect it can reduce the amount of video memory necessary to display the drawing, but the downside to this effect is it may increase the time it takes to load the images the first time that they are accessed and there is a reduction in the quality of the images when they are displayed in the viewport or plotted. To identify if Texture Compression is enabled, enter **3dconfig**, and click Manual Tune. Look at the Hardware Effects List in the Manual Performance Tuning dialog box.

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# Materials Window

## Quick Reference

Provides different panels of controls and settings to create, modify, and apply materials

Render
View ➤ Render ➤ Materials At the Command prompt, enter materials.
**materials**
Materials panel, Materials

Applies and modifies materials. The Materials window consists of different panel sections including Available Materials in Drawing Panel (page 795),Material Editor (page 797), Maps (page 799), Advanced Lighting Override (page 815), Material Scaling & Tiling (page 816), and Material Offset & Preview (page 819).

# Available Materials in Drawing

## Quick Reference

**materials**

Displays swatches of the materials that are available in the drawing. The default material is named Global. Click a swatch to select a material. The settings for that material are displayed in the Material Editor panel, and the swatch is outlined in yellow to indicate selection. One button above the swatches and two groups of buttons below it provide the following options. In addition, there are options available only on the shortcut menu.

**Toggle Display Mode** Switches the display of swatches from one swatch to rows of swatches. This button is located above the swatches in the upper-right corner.

**Swatch Geometry** Controls the type of geometry displayed for the selected swatch: box, cylinder, or sphere. The geometry changes in other swatches as you select them.

**Checkered Underlay Off/On** Displays a multicolored checkered underlay to help you visualize the degree of opacity of a material.

**Preview Swatch Lighting Model** Changes the lighting model from a single light source to a backlight lighting model. The selection from the fly out list changes the selected material swatch.

**Create New Material** Displays the Create New Material dialog box (page 823). After you enter a name, a new swatch is created to the right of the current swatch and the new swatch is selected.

**Purge from Drawing** Removes the selected material from the drawing. The Global material and any material that is in use cannot be deleted.

**Indicate Materials in Use** Updates the display of the in-use icon. Materials currently in use in the drawing display a drawing icon in the lower-right corner of the swatch.

**Apply Material to Objects** Applies the currently selected material to selected objects and faces.

**Remove Materials from Selected Objects** Detaches materials from the selected objects and faces.

**Select Objects with Material (Shortcut Menu Only)** Selects all objects in the drawing to which the selected material is applied. Faces with materials applied explicitly are not selected.

**Edit Name and Description (Shortcut Menu Only)** Opens the Edit Name and Description dialog box (page 823).

**Export to Active Tool Palette (Shortcut Menu Only)** Creates a material tool for the selected material on the currently active tool palette. If the Tool Palettes window is closed, it opens.

**Copy (Shortcut Menu Only)** Copies the selected material to the Clipboard. You can paste the material in the Tool Palettes window, or paste it back into the Available Materials panel as a copy.

**Paste (Shortcut Menu Only)** Pastes from the Clipboard either a swatch from the Available Materials panel, or a material tool from the Tool Palettes window.

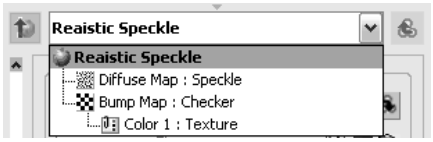
**Size (Shortcut Menu Only)** Controls the size of the swatches when they are displayed in rows.

# Nested Map Navigation

## Quick Reference

#### materials

Displays the nesting map navigation of the selected material swatch. Nested maps within a parent map or material can be selected. The Nested Map Navigation provides lists and buttons to control the current level of the nested maps.




**Home to Material Settings** Returns the display of nested map property settings back to the current basic material settings.

**Map name string** Displays in bold only the current map channel level and texture map or procedural map that is being edited. If you are at the top of the map channel hierarchy, the material name is displayed. If you are at the nested levels, the string in the display depend on where you are at the current

nested level. The display string will start with the Material name: current map channel - procedural Map | Sub-procedural map. For example: Material 1: Diffuse Map - Checker | Color 1 - Speckle | Color 2 | Noise.

**Custom drop-down tree control arrow** Displays levels of nested maps by selecting the drop-down arrow. By selecting a nested map the controls for that nested map will be displayed.

**Up One Level to Parent Map** Returns to the controls of the parent of the current sub-procedural map.

# Material Editor

## Quick Reference

#### materials

Edits the material selected in the Available Materials in Drawing panel. The name of the selected material is displayed following "Material Editor." The configuration of the Material Editor changes depending on the type of material and template that is selected.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Type** Specifies the type of material. Realistic and Realistic Metal are for materials based on physical qualities. Advanced and Advanced Metal are for materials with more options, including properties that you can use to create special effects; for example, simulated reflections.

**Template (Realistic and Realistic Metal Types)** Lists the templates available for the material type selected.

**Color (Realistic and Realistic Metal Types)** Displays the Select Color dialog box (page 251), where you specify the diffuse color of the material.

**By Object (Realistic and Realistic Metal Types)** Sets the color of the material based on the color of the object it is applied to.

**Ambient (Advanced and Advanced Metal Types)** Displays the Select Color dialog box (page 251), where you specify the color that appears on those faces lighted by ambient light alone.

**By Object (Advanced and Advanced Metal Types)** Sets the color of the material based on the color of the object it is applied to.

**First Lock Icon (Advanced and Advanced Metal Types)** When locked, the lock icon between Ambient and Diffuse sets the ambient color of the material to the diffuse color.

**Diffuse (Advanced and Advanced Metal Types)** Displays the Select Color dialog box (page 251), where you specify the diffuse color of the material. Diffuse color is the main color of the object.

**By Object (Advanced and Advanced Metal Types)** Sets the color of the material based on the color of the object it is applied to.

**Second Lock Icon (Advanced Type Only)** When locked, the lock icon between Diffuse and Specular sets the specular color of the material to the diffuse color.

**Specular (Advanced Type Only)** Displays the Select Color dialog box (page 251), where you specify the color of a highlight on a shiny material. The size of the highlight depends on the material's shininess.

**By Object (Advanced Type Only)** Sets the color of the material based on the color of the object it is attached to.

**Shininess** Sets the shininess of the material. The highlight on the face of a very shiny solid is smaller and brighter. A face that is less shiny reflects the light in more directions, creating a large, softer highlight.

**Opacity (Realistic and Advanced Types)** Sets the opacity of the material. A completely opaque solid object does not allow the passage of light through its surface. An object with no opacity is transparent.

**Reflection (Advanced and Advanced Metal Types)** Sets the reflectivity of the material. When set to 100, the material is fully reflective and the surrounding environment is reflected in the surface of any object to which the material is applied.

**Refraction Index (Realistic and Advanced Types)** Sets the refraction index of the material. Controls how light is refracted through an object with a partially transparent material attached. For example, at 1.0, the refraction index of air, the object behind the transparent object is not distorted. At 1.5, the object is distorted greatly, as if it were seen through a glass marble.

**Translucency (Realistic and Advanced Types)** Sets the translucency of the material. A translucent object transmits light, but light is also scattered within the object. The translucency value is a percentage: at 0.0, the material is not translucent; at 100.0, the material is as translucent as possible.

**Self-illumination** When set to a value more than 0, makes the object appear to give off light, independent of the lights in the drawing. When self-illumination is selected luminance is unavailable.

**Luminance (Realistic Type only)** Luminance is the value of light reflected off a surface. It is a measure of how bright or dark the surface is perceived. When selecting luminance the self-illumination is unavailable. Luminance is specified in real lighting units.

**Two Sided Material (Realistic type only)** When selected positive and negative face normals are rendered. When clear, only positive face normals are rendered. This setting is disabled if Force Two-Sided is set to "ON" in the Manage Render Presets dialog box. (Create Custom Render Presets.)

# Maps

## Quick Reference

**materials**

Assigns a pattern or texture to a material's diffuse color. The colors of the map replace the material's diffuse color in the Material Editor. For the Realistic and Realistic Metal material types, the Maps section of the Material window is divided into three mapping channel sections:Diffuse Map, Opacity Map, and Bump Map. For the Advanced and Advanced Metal material types, the Maps section is divided into four mapping channel sections: Diffuse Map, Reflection Map, Opacity Map, and Bump Map. Within each map channel a map type of Texture Map or Procedural Maps can be selected.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

# Diffuse Map Channel

## Quick Reference

**materials**

Diffuse mapping provides a pattern of colors for the material.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Diffuse Map** Makes the Diffuse map active on a material and available to be rendered.

**Diffuse Map Slider** Controls the blending of the image or procedural map with the assigned control properties for the material. Range = 0-100; Default = 100. When Texture Map is the map type, then no slider is displayed.

**Map Type** Specifies the type of map. You can use Texture Map to select an image file (for example, a tile pattern) or choose the Checker, Marble, Noise, Speckle, Tiles, Waves, or Wood map types.

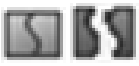
**Select Image** Displays the Select Image File dialog box, a standard file selection dialog box (page 931). After a file is selected, displays the name of the file is displayed.

The following formats can be used as texture maps:

- TGA (.tga)
- BMP (.bmp, .rle, .dib)
- PNG (.png)
- JFIF (.jpg, .jpeg)
- TIFF (.tif)
- GIF (.gif)
- PCX (.pcx)

 **Click for "Map Type" Settings** This button appears once a Texture Map image is selected or procedural map type is selected. Displays the controls for the Map Property Settings (page 805) The map control settings vary depending on the procedural map type selection.

 **Delete Map Information from Material** Removes the selected map information from the material.

 **Map Synchronize Toggle** When enabled (the button will appear as connected) the settings and value changes of the map channel are pushed

and synchronized to all map channels. When disabled (the button will appear as separated) the setting and value changes of the map channel are only relevant to the current map channel.


**Preview Map Channel Procedural Results** Displays the Diffuse Map Preview dialog box (page 824).

# Reflection Map Channel

## Quick Reference

#### materials

Controls how reflective the material is. Reflection maps (also known as environment maps) simulate a scene reflected on the surface of a shiny object.

**Reflection Map** Makes the Reflection map active on the material and available to be rendered.

**Reflection Map Slider** Controls the reflection of the map with the assigned control properties for the material. The higher the value, the more reflective the material. Range = 0-100; Default = 0. When Texture Map is the map type selected then no slider is displayed.

**Map Type** Specifies the type of map. You can use Texture Map to select an image file (for example, a tile pattern) or choose the Checker, Marble, Noise, Speckle, Tiles, Waves or Wood map types.

**Select Image** Displays the Select Image File dialog box, a standard file selection dialog box (page 931). After a file is selected, the name of the file is displayed.

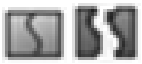
The following formats can be used as texture maps:

- TGA (.tga)
- BMP (.bmp, .rle, .dib)
- PNG (.png)
- JFIF (.jpg, .jpeg)
- TIFF (.tif)
- GIF (.gif)

- PCX (.pcx)

 **Click for "Map Type" Settings** This button appears once a Texture Map image is selected or procedural map type is selected. Displays the controls for the Map Property Settings (page 805). The map control settings vary depending on the procedural map type selection.

 **Delete Map Information from Material** Removes the selected map information from the material.

 **Map Synchronize Toggle** When enabled the settings and value changes of the map channel are pushed and synchronized to all map channels. When disabled the setting and value changes of the map channel are only relevant to the current map channel.

 **Preview Map Channel Procedural Results** Displays the Reflection Map Preview dialog box (page 824).

# Opacity Map Channel

## Quick Reference

**materials**

Defines which areas of the material should be transparent and which areas should not. Areas where the opacity map is applied appear transparent; other areas appear opaque. Areas of the material that are transparent still have a specular highlight (as though they were glass). If the texture is black and white the opacity will be transparent and opaque respectively.

**Opacity Map** Makes the Opacity map active on the material and available to be rendered.

**Opacity Map Slider** Controls solid appearance of the image or map. The lower the setting the more transparent the material is. The higher the setting the more solid the material is. Range = 0-100; Default = 100. When Texture Map is the map type, then no slider is displayed.

**Map Type** Specifies the type of map. You can use Texture Map to select an image file (for example, a tile pattern) or choose the Checker, Marble, Noise, Speckle, Tiles, Waves or Wood map types.

**Select Image** Displays the Select Image File dialog box, a standard file selection dialog box (page 931). After a file is selected, the name of the file is displayed.

The following formats can be used as texture maps:

- TGA (.tga)
- BMP (.bmp, .rle, .dib)
- PNG (.png)
- JFIF (.jpg, .jpeg)
- TIFF (.tif)
- GIF (.gif)
- PCX (.pcx)

 **Click for "Map Type" Settings** This button appears once a Texture Map image is selected or procedural map type is selected. Displays the controls for the Map Property Settings (page 805) The map control settings vary depending on the procedural map type selection.

 **Delete Map Information from Material** Removes the selected map information from the material.

 **Map Synchronize Toggle** When enabled the settings and value changes of the map channel are pushed and synchronized to all map channels. When disabled the setting and value changes of the map channel are only relevant to the current map channel.

 **Preview Map Channel Procedural Results** Displays the Opacity Map Preview dialog box (page 824).

# Bump Map Channel

## Quick Reference

**materials**

Bump mapping makes an object appear to have a bumpy or irregular surface.You can select an image file or Procedural Maps to use for bump mapping. Use bump maps when you want to take the smoothness off a surface, or to create an embossed look.

**Bump Map** Makes the Bump map active on the material and available to be rendered.

**Bump Slider** Controls the bumpiness of the texture. Range = -1000 to 1000; Default = 1000.0. When Texture Map is the map type selected, then no slider is displayed.

**Map Type** Specifies the type of map. You can use Texture Map to select an image file (for example, a tile pattern) or choose the Checker, Marble, Noise, Speckle, Tiles, Waves or Wood map types.

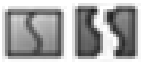
**Select Image** Displays the Select Image File dialog box, a standard file selection dialog box (page 931). After a file is selected, the name of the file is displayed.

The following formats can be used as texture maps:

- TGA (.tga)
- BMP (.bmp, .rle, .dib)
- PNG (.png)
- JFIF (.jpg, .jpeg)
- TIFF (.tif)
- GIF (.gif)
- PCX (.pcx)

 **Click for "Map Type" Settings** This button appears once a Texture Map image is selected or procedural map type is selected. Displays the controls for the Map Property Settings (page 805) The map control settings vary depending on the procedural map type selection.

 **Delete Map Information from Material** Removes the selected map information from the material.

 **Map Synchronize Toggle** When enabled the settings and value changes of the map channel are pushed and synchronized to all map channels. When disabled the setting and value changes of the map channel are only relevant or to the current map channel.

 **Preview Map Channel Procedural Results** Displays the Bump Map Preview dialog box (page 824).

# Map Property Settings

## Quick Reference

#### materials

For each map type there are different property settings. See each map type for specific settings:

- Texture Map (page 806)
- Checker (page 806)
- Marble (page 807)
- Noise (page 808)
- Speckle (page 809)
- Tiles (page 810)
- Waves (page 813)
- Wood (page 814)

## Texture Map Property Settings

### Quick Reference

**materials**

Controls the property settings for Texture Map.

**Click for Texture Map Settings** Displays a window with property settings to adjust the appearance of the texture. This button appears once an image is selected for the Texture Map.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Texture Map Slider** Controls the blending of the texture map with the assigned control properties for the material.

**Scaling & Tiling** Provides settings (page 817) to adjust the scaling and tiling for the map.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

## Checker Property Settings

### Quick Reference

**materials**

Controls the property settings for the Checker procedural map.

**Click for Checker Settings** Displays a window with property settings to adjust the appearance of the Checker procedural map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Color 1** Provides the option of selecting a color or a sub-procedural map for one of the checkers.

- **Map Type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural

map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears.

- **Color.** Click the color box to display the Select Color dialog box (page 251).

**Swaps the Map Types** Swaps the Map Types between Color 1 and Color 2.

**Color 2** Provides the option of selecting a color or a sub-procedural map for one of the checkers.

- **Map Type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears.
- **Color.** Click the color box to display the Select Color dialog box (page 251).

**Soften** Adjusts the value in the amount of softness or blurring between the edge of the two colors or map types. Higher values blur more. A value of 0.0 indicates sharp edges. Range = 0 to 5.00; Default = 0.

**Scaling & Tiling** Provides settings (page 817) to adjust the scaling and tiling for the map.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Marble Property Settings

## Quick Reference

#### materials

Controls the property settings for the Marble procedural map.

**Click for Marble Settings** Displays a window with property settings to adjust the appearance of the Marble procedural map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Stone color** Specifies the color of the stone. Displays the Select Color dialog box (page 251).

**Swaps the Colors** Swaps the colors between the Stone and Vein color.

**Vein color** Specifies the vein color of the marble. Displays the Select Color dialog box (page 251).

**Vein spacing** Sets the space between veins. Range = 0-100; Default = 1.00.

**Vein width** Sets the width of the veins. Range = 0-100; Default = 1.00.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Noise Property Settings

## Quick Reference

**materials**

Controls the property settings for the Noise procedural map.

**Click for Noise Settings** Displays a window with property settings to adjust the appearance of the Noise procedural map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Noise Type** Creates random perturbation of a surface based on the interaction of two colors.

**Regular**. Generates plain noise. Basically the same as fractal noise with levels setting at 1. When the noise type is set to Regular, the Levels spinner is inactive (because Regular is not a fractal function).

**Fractal**. Generates noise using a fractal algorithm. The Levels option sets the number of iterations for fractal noise.

**Turbulence.** Generates fractal noise with an absolute value function applied to it to make fault lines. Note that the noise amount must be greater than 0 to see any effects of turbulence.

**Size.** Sets the scale of the noise function. Smaller values give smaller chunks of noise.

**Color 1** Provides the option of selecting a color or a sub-procedural map for one of the checkers.

- **Map Type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears.
- **Color.** Click the color box to display the Select Color dialog box (page 251).

**Swaps the Map Types** Swaps the map types between Color 1 and Color 2.

**Color 2** Provides the option of selecting a color or a sub-procedural map for one of the checkers.

- **Map Type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears
- **Color.** Click the color box to display the Select Color dialog box (page 251).

**Noise threshold** Specifies a noise threshold. the settings are:

- **Low.** Sets the low threshold. Range = 0 to 1; Default = 0.
- **High.** Sets the high threshold. Range = 0 to 1; Default = 1.
- **Level.** Determines how much fractal energy is used for the Fractal and Turbulence noise functions. You can set the exact amount of turbulence you want and also animate the number of fractal levels. Default = 3.
- **Phase.** Controls the appearance of the noise. Use this option to vary the appearances of different objects that use the noise map.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Speckle Property Settings

## Quick Reference

#### materials

Controls the property settings for the Speckle procedural map.

**Click for Speckle Settings** Displays a window with property settings to adjust the appearance of the Speckle procedural Map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Color 1** Specifies a color for the speckle. Displays the Select Color dialog box (page 251).

**Swaps the Colors** Swaps the colors between Color 1 and Color 2.

**Color 2** Specifies a second color for the speckle. Displays the Select Color dialog box (page 251).

**Size** Adjusts the size of the speckles.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Tiles Property Settings

## Quick Reference

**materials**

Controls the property settings for the Tiles procedural map.

**Click for Tiles Settings** Displays a window with property settings to adjust the appearance of the Tiles procedural map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Pattern type** Determines either a preset or custom tile pattern. Preset patterns typically disable most or all of the stacking layout and row/column edit controls.

- **Custom Tile**.
- **Running Bond.**
- **Common Flemish Bond.**
- **English Bond.**

- **1/2 Running Bond.**
- **Stack Bond.**
- **Fine Running Bond.**
- **Fine Stack Bond.**

**Random seed** Randomly applies patterns of color variation to the tiles. Does not require any other setting to generate completely different patterns.

**Tiles Setup** Provides the following options for mapping with Tiles:

- **Map type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears.
- **Select Image.** If Texture Map is selected as the map type, specifies the image to use.
- **Color.** Click the color box to display the Select Color dialog box (page 251).
- **Go to 'map type" Settings.** If Texture Map or a sub-procedural map is selected, displays the property settings.
- **Horizontal count.** Controls the number of tiles in a row. Default = 4.
- **Vertical count.** Controls the number of tiles in a column. Default = 4.
- **Color variance.** Controls the color variation of the tiles. Default = 0.05.
- **Fade variance.** Controls the fading variation of the tiles.

**Swaps the Map Types** Swaps the map types between the Tiles and the Grout.

**Grout setup** Controls the appearance of the grout.

- **Map type.** Specifies Texture Map, sub-procedural map type, or color. If Texture Map is selected, select an image. If Texture Map or a sub-procedural map is selected, select the Click for "map type" Settings button. If Solid color is selected then a color selection box appears.
- **Select Image.** If Texture Map is selected as the map type, specifies the image to use.
- **Color.** Click the color box to display the Select Color dialog box (page 251).
- **Go to 'map type" Settings.** If Texture Map or a sub-procedural map is selected, displays the property settings.

- **Horizontal gap.** Controls the horizontal size of the grout between the tiles. This value is locked by default to the vertical gap, so that both values change as you edit one or the other. Range = 0 to 100; Default = 0.50.
- **Vertical gap.** Controls the vertical size of the grout between the tiles. This value is locked by default to the horizontal gap, so that both values changes as you edit one or the other. Range = 0 to 100; Default = 0.50.
- **Lock.** Locks and unlocks the horizontal and vertical grout gaps.
- **Rough.** Controls the roughness of the edges of the mortar. Range = 0 to 200; Default = 0.

**Stacking Layout** Specifies the controls when Custom Tiles is selected.

- **Line shift.** Shifts every second row of tiles a distance of one unit. Range = 0 to 100; Default = 0.50.
- **Random shift.** Randomly shifts all rows of tiles a distance of one unit. Range = 0 to 100; Default = 0.

**Row Modify** For Custom Tiles, creates a custom pattern for rows based upon the values of Per Row and Change settings. Default = Off.

- **Selection box.** Turns the settings for Grout Row Modify on and off.
- **Per row.** Specifies which rows to change. When Per Row equals 0, no rows change. When Per Row equals 1, every row changes. When Per Row is a value greater than 1, the change appears every *N* rows: a value of 2 changes every second row, a value of three changes every third row, and so on. Range = 0 to 5. Default = 1.
- **Change.** Changes the width of tiles in the affected rows. A value of 1.0 is the default tile width. Values greater than 1.0 increase the width of tiles, and values less than 1.0 decrease it. Range = 0.0 to 5.0. A value of 0 is a special case: when the change value is 0.0, no tiles appear in that row and the underlying material shows through. Default = 1.

**Column Modify** For Custom Tiles, creates a custom pattern for columns based upon the values of Per Column and Change settings. Default = Off.

- **Selection box.** Turns the settings for Grout Column Modify on and off.
- **Per column.** Specifies which columns to change. When Per Column equals 0, no columns change. When Per Column equals 1, every column changes. When Per Column is a value greater than 1, the change appears every *N*

columns: a value of 2 changes every second column, a value of three changes every third column, and so on. Range = 0 to 50. Default = 1.

- **Change.** Changes the height of tiles in the affected rows. A value of 1.0 is the default tile width. Values greater than 1.0 increase the height of tiles, and values less than 1.0 decrease it. Range = 0.0 to 5.0. A value of 0 is a special case: when the change value is 0.0, no tiles appear in that column and the underlying material shows through. Default = 1.

**Offset & Preview** Provides settings (page 820) to adjust the scaling and tiling for the map.

**Scaling & Tiling** Provides settings (page 817) to adjust the material offset and preview for the map.

# Waves Property Settings

## Quick Reference

**materials**

Controls the property settings for the Waves. procedural map.

**Click for Waves Settings** Displays a window with property settings to adjust the appearance of the Waves procedural Map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Color 1** Specifies a color for Waves. The Select Color dialog box (page 251) is displayed.

**Swaps the Colors** Swaps the colors between Color 1 and Color 2.

**Color 2** Specifies a second color for Waves. The Select Color dialog box (page 251) is displayed.

**Distribution** Specifies the distribution of the waves. 3D distributes the wave centers on the surface on an imaginary sphere, affecting all sides of a 3D object. 2D distributes the wave in circles centered on the *XY* plane, which is more appropriate for flat water surfaces such as oceans and lakes.

**Number of waves** Specifies how many wave sets are used in the pattern. Wave sets are groups of radially symmetrical waves that originate from randomly

computed points along the surface of an imaginary sphere inside the object (a circle, in the case of 2D wave distribution). For calm water, set this to a low number. Use a high number for choppy water. Range = 1 to 50; Default = 3.

**Wave radius** Specifies the radius of the imaginary sphere (3D distribution) or circle (2D distribution) whose surface is the origin of each wave set. A large radius produces large circular wave patterns, while a small radius produces dense, smaller waves. Default = 1000.

**Len Min** Defines the minimum interval for each wave center. If values are close together, the waves appear more regular; if they are farther apart, the waves are less regular. Default = 50.

**Len Max** Defines the maximum interval for each wave center. If values are close together, the waves appear more regular; if they are farther apart, the waves are less regular. Default = 50.

**Amplitude** Specifies the amplitudes of the waves. The 3D setting distributes the wave centers on the surface on an imaginary sphere, affecting all sides of a 3D object. The 2D setting distributes the wave in circles centered on the *XY* plane, which is more appropriate for flat water surfaces such as oceans and lakes. Range = 0 to 10000; default = 1.

**Phase** Shifts the wave pattern. Range = 0 to 1000; Default = 0.

**Random seed** Provides a seed number to generate the water pattern. The pattern changes with each seed. Range = 0 to 65535; default = 30159.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Wood Property Settings

## Quick Reference

#### materials

Controls the property settings for the Wood procedural map.

**Click for Wood Settings** Displays a window with property settings to adjust the appearance of the Wood procedural map. The Nested Map Navigation (page 796) displays the current nested map level.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Color 1** Specifies a color for the wood. The Select Color dialog box (page 251) is displayed.

**Swaps the Colors** Swaps the colors between Color 1 and Color 2.

**Color 2** Specifies a second color the wood. The Select Color dialog box (page 251) is displayed.

**Radial noise** Sets the relative randomness of the radial noise pattern on a plane perpendicular to the grain on the wood. Range = 0 to 100; Default 1.00.

**Axial noise** Sets the relative randomness of the axial noise pattern on a plane parallel with the grain on the wood. Range = 0 to 100; Default 1.00.

**Grain thickness** Sets the relative thickness of the color bands that make up the grain on the wood. Range = 0 to 100; Default 0.50.

**Offset & Preview** Provides settings (page 820) to adjust the material offset and preview for the map.

# Advanced Lighting Override

## Quick Reference

#### materials

Sets parameters to affect the rendering of a material when it is lit by indirect illumination from global illumination and/or final gather. The advanced lighting override provides controls to change the properties of the material to affect the rendered scene. This control is only available on Realistic & Realistic Metal material types.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Color Bleed Scale Slider** Increases or decreases the saturation of reflected color. Range = 0.0 to 1000; Default = 100.

**Indirect Bump Scale Slider** Scales the effect of the base material's Bump map in areas lit by indirect light. When this value is zero, no bump mapping is done for indirect light. Increasing the scale increases the bump effect under indirect lighting. This value does not affect the Bump amount in areas where the base material is lit directly. Range 0 to 1000; Default = 100.

**Reflectance Scale Slider** Increases or decreases the amount of energy the material reflects. Reflectance is the percentage of diffuse light energy that is reflected from a material. Range = 0 to 1000; Default = 100.

**Transmittance Scale Slider** Increases or decreases the amount of energy the material transmits.Transmittance is the amount of light energy transmitted through a material. A completely opaque material has 0% transmittance. Range = 0 to 1000; Default = 100.

# Material Scaling & Tiling

## Quick Reference

**materials**

Specifies scaling and tiling of maps on materials.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Scale units** Specifies the units to use in scaling.

- **None.** Specifies a fixed scale.
- **Fit to Gizmo.** Fits the image to the face or object.
- **Units.** Specifies the type of units to scale in real-world units (Millimeters, Centimeters, Meters, Kilometers, Inches, Feet, Survey Feet, Miles, or Yards).

**U Tile** Adjusts the tiling of the image along the U axis. Selecting None, Tile, and Mirror specifies the type of tiling for the material in the U axis. The preview is updated as you change the value.

- **None.** Controls the map to not be tiled within the material; just one tile will appear across the object it is applied to.
- **Tile.** Controls the map to be tiled within the material. This affects real-world scale.
- **Mirror.** Controls the map to be tiled, but each tile will be a mirror of the adjacent one.

UVW coordinates are similar to XYZ coordinates, but they refer to the image and move with it when it moves or rotates.

**U Tile Spinner** Adjusts, for U Tile selections of Tile and Mirror, the value in the tiling amount on the U axis for the image. Range = 1 to 500. When the values are adjusted via the spinner, the U Tile trackbar is updated to reflect the amount and is displayed in the interactive preview in the Material Offset & Preview section.

**V Tile** Adjusts the tiling of the image along the V axis. Selecting None, Tile, and Mirror specifies the type of tiling for the material in the V axis. The preview is updated as you change the value.

- **None.** Controls the map to not be tiled within the material; just one tile will appear across the object it is applied to.
- **Tile.** Controls the map to be tiled within the material. This affects real-world scale.
- **Mirror.** Controls the map to be tiled, but each tile will be a mirror of the adjacent one.

UVW coordinates are similar to XYZ coordinates, but they refer to the image and move with it when it moves or rotates.

**V Tile Spinner** Adjusts, for V Tile selections of Tile and Mirror, the value of the tiling amount on the V axis for the image. Range = 1 to 500. When the values are adjusted via the spinner, the V Tile trackbar is updated to reflect the amount and is displayed in the interactive preview in the Material Offset & Preview section.

**Lock Aspect Ratio** Locks the shape of the map. The original U Tile and V Tile values are based on the aspect ratio of the map. The greater of the two is assigned a value of 1, and the other is assigned a value that maintains the ratio. When the length or width value is changed, the other value changes as needed to maintain the shape.

## Material Scaling & Tiling for Sub-procedural Maps

### Quick Reference

#### materials

Specifies scaling and tiling for sub-procedural maps on materials. This functionality is only available on Texture map and 2D sub-procedural maps (Checker, and Tiles). This control is not available on 3D sub-procedural maps (Noise, Speckle, Marble, Wood and Wave).

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Map Synchronize** When enabled the settings and value changes of the map channel are pushed and synchronized to all map channels. When disabled the setting and value changes of the map channel are only relevant or to the current map channel. This control is available within the sub-procedural material Scaling & Tiling control. It is not available in the main material window.

**Scale units** Specifies the units to use in scaling.:

- **None.** Specifies a fixed scale.
- **Fit to Gizmo.** Fits the image to the face or object.

**U Tile** Adjusts the tiling of the image along the U axis. Selecting None, Tile, and Mirror specifies the type of tiling for the material in the U axis. The preview is updated as you change the value.

- **None.** Controls the map to not be tiled within the material; just one tile will appear across the object it is applied to.
- **Tile.** Controls the map to be tiled within the material. This affects real-world scale.
- **Mirror.** Controls the map to be tiled, but each tile will be a mirror of the adjacent one.

UVW coordinates are similar to XYZ coordinates, but they refer to the image and move with it when it moves or rotates.

**U Tile Spinner** Adjusts, for U Tile selections of Tile and Mirror, the value in the tiling amount on the U axis. Range = 1 to 500. When the values are adjusted via the spinner, the U Tile trackbar is updated to reflect the amount and is displayed in the interactive preview in the Material Offset & Preview section.

**V Tile** Adjusts the tiling of the image along the V axis. Selecting None, Tile, and Mirror specifies the type of tiling for the material in the V axis. The preview is updated as you change the value.

- **None.** Controls the map to not be tiled within the material; just one tile will appear across the object it is applied to.
- **Tile.** Controls the map to be tiled within the material. This affects real-world scale.

- **Mirror.** Controls the map to be tiled, but each tile will be a mirror of the adjacent one.

UVW coordinates are similar to XYZ coordinates, but they refer to the image and move with it when it moves or rotates.

**V Tile Spinner** Adjusts, for V Tile selections of Tile and Mirror, the value of the tiling amount on the V axis. Range = 1 to 500. When the values are adjusted via the spinner, the V Tile trackbar is updated to reflect the amount and is displayed in the interactive preview in the Material Offset & Preview section.

**Lock Aspect Ratio** Locks the shape of the map. The original U Tile and V Tile values are based on the aspect ratio of the map. The greater of the two is assigned a value of 1, and the other is assigned a value that maintains the ratio. When the length or width value is changed, the other value changes as needed to maintain the shape.

# Material Offset & Preview

## Quick Reference

**materials**

Specifies the offset and preview properties of maps on materials.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Auto-regen** Specifies that any changes in a material are reflected in the interactive swatch (preview) right away. When cleared, any changes made to the map channel are not reflected in the interactive swatch.

**Update** Updates the interactive swatch preview.

**Preview size** Zooms in or out of the preview.

**Preview** Displays a preview of the map on the face or object that updates as you change settings. Click and drag inside the square to adjust the U and V offsets.

When Scale is selected, the square represents a 1-unit by 1-unit tile. If Units is set to None, the unit is whatever the units are in the drawing. When Fit to Gizmo is selected, the square represents the tile to be stretched to fit the face of the object.

**U Tile Slider** Adjusts the tiling of the map along the U axis. The preview is updated as you change the value. The slider is only available on Texture Map and 2D procedurals (Checker, and Tiles). The slider is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood)..

**V Tile Slider** Adjusts the tiling of the map along the V axis. The preview is updated as you change the value. The slider is only available on Texture Map and 2D procedurals (Checker, and Tiles). The slider is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood).

**Lock** Locks the shape of the map. When the length or width value is changed, the other value changes as needed to maintain the shape. The lock is only available on Texture Map and 2D procedurals (Checker, and Tiles). The lock is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood).

**U offset** Moves the starting point of the map along the U axis. You can set this value interactively by moving the square inside the preview.

**V offset** Moves the starting point of the map along the V axis. You can set this value interactively by moving the square inside the preview.

**Rotation** Rotates the map around the W axis of the UVW coordinate system. Rotation is not available for spherical and cylindrical mapping. Use *MATERIALMAP* to display the mapping gizmo that can rotate box, planar, spherical and cylindrical maps.

## Material Offset & Preview for Sub-procedural Maps

### Quick Reference

#### materials

Specifies the Offset & Preview properties of maps on materials.

**Collapse/Expand Display Panel** Collapses (up arrows) and expands (down arrows) the display panel.

**Map Synchronize Toggle Button** When enabled the settings and value changes of the map channel are pushed and synchronized to all map channels. When disabled the setting and value changes of the map channel are only relevant or to the current map channel.

**Auto-regen** Specifies that any changes in a material are reflected in the interactive swatch (preview) right away. When cleared, any changes made to the map channel are not reflected the interactive swatch.

**Update** Updates the interactive swatch preview.

**Preview size** Zooms in or out of the preview.

**Preview** Displays a preview of the map on the face or object that updates as you change settings. Click and drag inside the square to adjust the U and V offsets.

If Scale units is set to None in Scaling & Tiling, the unit is whatever the units are in the drawing. The preview square represents a 1-unit by 1-unit tile. When Fit to Gizmo is selected, the preview square represents the tile to be stretched to fit the face of the object.

**U Tile Slider** Adjusts the tiling of the map along the U axis. The preview is updated as you change the value The slider is only available on Texture Map and 2D procedurals (Checker, and Tiles). The slider is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood).

**V Tile Slider** Adjusts the tiling of the map along the V axis. The preview is updated as you change the value. The slider is only available on Texture Map and 2D procedurals (Checker, and Tiles). The slider is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood).

**Lock** Locks the shape of the map. When the length or width value is changed, the other value changes as needed to maintain the shape. The lock is only available on Texture Map and 2D procedurals (Checker, and Tiles). The Lock is not available on 3D procedurals (Noise, Speckle, Marble, Waves or Wood).

**U offset** Moves the starting point of the map along the U axis. You can set this value interactively by moving the square inside the preview.

**V offset** Moves the starting point of the map along the V axis. You can set this value interactively by moving the square inside the preview.

**Rotation** Rotates the map around the W axis of the UVW coordinate system. Rotation is not available for spherical and cylindrical mapping. Use *MATERIALMAP* to display the mapping gizmo that can rotate box, planar, spherical and cylindrical maps.

# Materials Tool Palette

## Quick Reference

**materials**

Displays swatches of the materials that are available in the drawing. The default material is named Global. Click a swatch to select a material. The settings for that material are displayed in the Material Editor panel, and the swatch is outlined in yellow to indicate selection. One button above the swatches and two groups of buttons below it provide the following options. In addition, there are options available only on the shortcut menu.

**Toggle Display Mode** Switches the display of swatches from one swatch to rows of swatches. This button is located above the swatches in the upper right corner.

**Swatch Geometry** Controls the type of geometry displayed for the selected swatch: box, cylinder, or sphere. The geometry changes in other swatches as you select them.

**Checkered Underlay Off/On** Displays a multicolored checkered underlay to help you visualize the degree of opacity of a material.

**Preview Swatch Lighting Model** Changes the lighting model from a single light source to a backlight lighting model. The selection from the flyout changes the selected material swatch.

**Create New Material** Displays the Create New Material dialog box (page 823). After you enter a name, a new swatch is created to the right of the current swatch and the new swatch is selected.

**Purge from Drawing** Removes the selected material from the drawing. The Global material and any material that is in use cannot be deleted.

**Indicate Materials in Use** Updates the display of the in-use icon. Materials currently in use in the drawing display a drawing icon in the lower-right corner of the swatch.

**Apply Material to Objects** Applies the currently selected material to objects and faces.

**Remove Materials from Selected Objects** Detaches materials from the selected objects and faces.

**Select Objects with Material (Shortcut Menu Only)** Selects all objects in the drawing to which the selected material is applied. Faces with materials applied explicitly are not selected.

**Edit Name and Description (Shortcut Menu Only)** Opens the Edit Name and Description dialog box (page 823).

**Export to Active Tool Palette (Shortcut Menu Only)** Creates a material tool for the selected material on the currently active tool palette. If the Tool Palettes window is closed, it opens.

**Copy (Shortcut Menu Only)** Copies the selected material to the Clipboard. You can paste the material in the Tool Palettes window, or paste it back into the Available Materials panel as a copy.

**Paste (Shortcut Menu Only)** Pastes from the Clipboard either a swatch from the Available Materials panel, or a material tool from the Tool Palettes window.

**Size (Shortcut Menu Only)** Controls the size of the swatches when they are displayed in rows.

# Material Tool Property Editor

## Quick Reference

**materials**

Controls the properties of the selected material tool. The properties displayed here are described in the Material Editor section of the Materials window (page 794). A few properties, such as swatch geometry and checkered underlay, are not available in the Material Tool Property Editor.

When you edit a material tool, only the tool itself is affected. If the material is being used in the drawing and you want to apply the changes, use the material tool to reapply the changed material to the objects.

To make the changed material available in the drawing, apply it to objects in the drawing or copy it to the Materials window.

Unlike the Properties palette, the Material Tool Property Editor must be closed before you can continue working in the drawing.

# Create New Material and Edit Name and Description Dialog Boxes

## Quick Reference

**materials**

Names materials and provides space for a description.

**Name** Names the material.

**Description** Provides an optional description for the material.

## Map Preview Dialog Boxes

### Quick Reference

**materials**

Provides a larger more detailed view of the selected map channel. The preview window can remain open while making changes to the map property settings. The settings can be previewed as they are made at the current map level or any other level. The preview is a 2D representation of the texture.

**Auto-regen** Specifies that the preview swatch is updated automatically when changes are made to the material settings.

**Update** Specifies that changes to the preview can be done manually. If Auto-regen is clear, and when changes are made to the map property settings, the Update button becomes active.

# MATERIALSCLOSE

### Quick Reference

Closes the Materials window

**materialsclose**

The Materials window (page 794) closes.

# MATLIB

### Quick Reference

Obsolete

**matlib**

The Materials window (page 794) is displayed.

# MEASURE

## Quick Reference

Places point objects or blocks at measured intervals on an object

Draw ➤ Point ➤ MeasureAt the Command prompt, enter measure.
**measure**
Select object to measure:
Specify length of segment (page 825) or [Block (page 826)]: *Specify a distance or enter* **b**

The points or blocks drawn by MEASURE are placed in the Previous selection set, so you can choose them all by entering **p** at the next Select Objects prompt. You can use the Node object snap to draw an object by snapping to the point objects. You can then remove the points by entering **erase previous**.

The markers are placed in the user coordinate system (UCS) of the object being measured (except for 3D polylines in the current UCS). Markers are always placed on the object, regardless of the elevation settings.

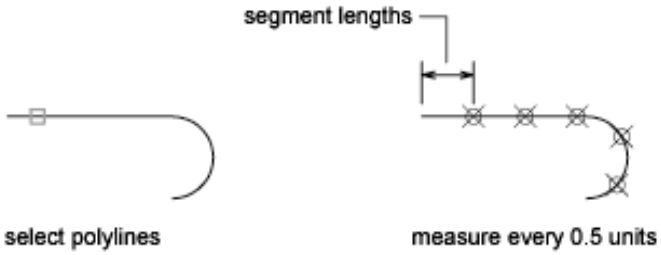
If you use point objects for the markers, you can make the points easier to see by changing their appearance with the *PDMODE* system variable.

**Length of Segment** Places point objects at the specified interval along the selected object, starting at the endpoint closest to the point you used to select the object.

Measurement of closed polylines starts at their initial vertex (the first one drawn).

Measurement of circles starts at the angle from the center set as the current snap rotation angle. If the snap rotation angle is 0, then the measurement of the circle starts to the right of center, on its circumference.


The illustration shows how MEASURE marks 0.5-unit distances along a polyline, with the *PDMODE* system variable set to 35.




**Block** Places blocks at a specified interval along the selected object.

Enter name of block to insert: *Enter the name of a block currently defined in the drawing*

Align block with object? [Yes/No] <Y>: *Enter* **y** *or* **n** *or press* ENTER

If you enter **y**, the block is rotated about its insertion point so that its horizontal lines are aligned with, and drawn tangent to, the object being measured. If you enter **n**, the block is always inserted with a 0 rotation angle.

Specify length of segment:

After you specify the segment length, the block is inserted at the specified interval. If the block has variable attributes, these attributes are not included.

# MENU

## Quick Reference

Loads a customization file

**menu**

The Select Customization File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter or select a customization file name. When you press ENTER or choose Open, the named file is loaded.

A customization file is an XML-based file containing the menus, toolbars, workspaces, and other interface elements that you can customize.

You can create a customized menu file and use MENU or CUI to load the file.

**NOTE** The Legacy Menu File (MNS) and Legacy Menu Template (MNU) files used in past releases have been replaced with just one file type, the XML-based CUI file.

# MENULOAD

## Quick Reference

Obsolete

**menuload**

The Load/Unload Customizations dialog box (page 285) is displayed. (CUILOAD (page 285) command)

**NOTE** The Legacy Menu File (MNS) and Legacy Menu Template (MNU) files used in past releases have been replaced with just one file type, the XML-based CUI file.

# MENUUNLOAD

## Quick Reference

Obsolete

**menuunload**

The Load/Unload Customizations dialog box (page 285) is displayed. (CUIUNLOAD (page 286) command)

**NOTE** The Legacy Menu File (MNS) and Legacy Menu Template (MNU) files used in past releases have been replaced with just one file type, the XML-based CUI file.

# MINSERT

## Quick Reference

Inserts multiple instances of a block in a rectangular array

**minsert**
Enter block name or [?]: *Enter a name, enter* **?** *to list the currently defined blocks in the drawing, or enter ~ to display the Select Drawing File dialog box*

**NOTE** You cannot precede the name of a block with an asterisk to explode the block's objects during insertion, as you can with INSERT (page 669).

Specify insertion point (page 828) or [Scale (page 830)/X (page 830)/Y (page 831)/Z (page 831)/Rotate (page 832)/PScale (page 833)/PX (page 833)/PY (page 834)/PZ (page 834)/PRotate (page 834)]*: Specify a point or enter an option*

Options at the insertion point preset the scale and rotation of a block before you specify its position. Presetting is useful for dragging a block using a scale factor and a rotation other than 1 or 0. If you enter one of the options, respond to the prompts by specifying a distance for the scale options or an angle for rotation.

Blocks inserted using MINSERT cannot be exploded.

You cannot use MINSERT with blocks.

### Insertion Point

Specifies a location for the blocks.

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER
**X Scale Factor** Sets *X* and *Y* scale factors.

Enter Y scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | | ): *Enter a value or specify a distance*

**Corner** Sets the scale factor by using the block insertion point and the opposite corner.

Specify opposite corner: *Specify a point*

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | | ): *Enter a value or specify a distance*

**XYZ** Sets *X*, *Y*, and *Z* scale factors.

Specify X scale factor or [Corner] <1>: *Enter a value or press* ENTER *to use the same scale factor*

If you enter **c**, you specify a corner point. The specified point and the block insertion point determine the *X* and *Y* scale factors.

Specify Y scale factor or <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter Z scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | | ): *Enter a value or specify a distance*

### Scale

Sets the scale factor for the *X*, *Y*, and *Z* axes. The scale for the *Z* axis is the absolute value of the specified scale factor.

Specify scale factor for XYZ axes: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*
Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one row, the following prompt is displayed:

Specify distance between columns ( | | | ): *Enter a value or specify a distance*

### X

Sets the *X* scale factor.

Specify X scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*
Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | ): *Enter a value or specify a distance*

### Y

Sets the *Y* scale factor.

Specify Y scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*
Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | ): *Enter a value or specify a distance*

### Z

Sets the *Z* scale factor.

Specify Z scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*
Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | ): *Enter a value or specify a distance*

### Rotate

Sets the angle of insertion for both the individual blocks and the entire array.

Specify rotation angle <0>:
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option or press* ENTER
**X Scale Factor** Sets the *X* scale factor.

Enter Y scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | ): *Enter a value or specify a distance*

**Corner** Sets the scale factor by the specified point and the block insertion point.

Specify opposite corner: *Specify a point*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | ): *Enter a value or specify a distance*

**XYZ** Specify X scale factor or [Corner] <1>: *Specify a nonzero value, enter* **c**, *or press* ENTER

You can determine the *X* and *Y* scale factors by entering a scale factor value or by specifying a corner point. If you specify a corner point, the specified point and the block insertion point determine the *X* and *Y* scale factors for the block.

Enter Z scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns ( | | | ): *Enter a value or specify a distance*

### PScale

Sets the scale factor for the *X*, *Y*, and *Z* axes to control the display of the block as it is dragged into position.

Specify preview scale factor for XYZ axes: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

### PX

Sets the scale factor for the *X* axis to control the display of the block as it is dragged into position.

Specify preview X scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

#### PY

Sets the scale factor for the *Y* axis to control the display of the block as it is dragged into position.

Specify preview Y scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

#### PZ

Sets the scale factor for the *Z* axis to control the display of the block as it is dragged into position.

Specify preview Z scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

#### PRotate

Sets the rotation angle of the block as it is dragged into position.

Specify preview rotation angle:
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

# MIRROR

## Quick Reference

Creates a mirror image copy of objects

Modify 

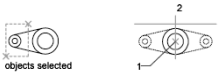
Modify ➤ MirrorDoes not exist in the menus.

**mirror**

Select objects: *Use an object selection method and press* ENTER *to finish*

Specify first point of mirror line: *Specify a point*

Specify second point of mirror line: *Specify a point*




The two specified points become the endpoints of a line about which the selected objects are mirrored. For mirroring in 3D, this line defines a mirroring plane perpendicular to the *XY* plane of the user coordinate system (UCS) containing the mirror line.

Erase source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

**Yes** Places the mirrored image into the drawing and erases the original objects.

original objects deleted

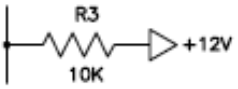
**No** Places the mirrored image into the drawing and retains the original objects.

original objects retained

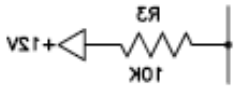
---

**NOTE** By default, when you mirror a text object, the direction of the text is not changed. Set the *MIRRTEXT* system variable to 1 if you do want the text to be reversed.

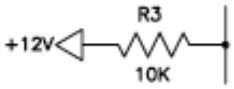
---




before mirroring



after mirroring
MIRRTEXT=1




after mirroring
MIRRTEXT=0

# MIRROR3D

## Quick Reference

Creates a mirror image of objects about a plane

**mirror3d**

It is recommended that you use the grip tools available through the *3DMOVE* and *3DROTATE* commands to manipulate 3D objects. For more information about using grip tools, see Use Grip Tools to Modify Objects.

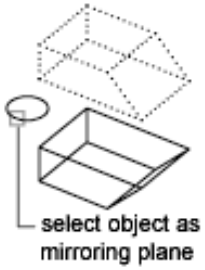
Select objects: *Use an object selection method and press* ENTER *to finish*
Specify first point of mirror plane (3 points) or [Object (page 836)/Last (page 836)/Zaxis (page 837)/View (page 837)/XY/YZ/ZX (page 837)/3points (page 838)] <3points>: *Enter an option, specify a point, or press* ENTER

**Object** Uses the plane of a selected planar object as the mirroring plane.

Select a circle, arc, or 2D-polyline segment:

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object is placed into the drawing and the original objects are retained.




**Last** Mirrors the selected objects about the last defined mirroring plane.

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object are placed into the drawing and the original objects are retained.

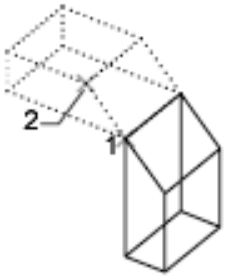
**Z Axis** Defines the mirroring plane by a point on the plane and a point normal to the plane.

Specify point on mirror plane: *Specify a point (1)*

Specify point on Z-axis (normal) of mirror plane: *Specify a point (2)*

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object are placed into the drawing and the original objects are retained.




**View** Aligns the mirroring plane with the viewing plane of the current viewport through a point.

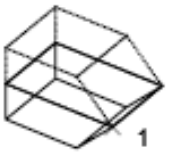
Specify point on view plane <0,0,0>: *Specify a point or press* ENTER

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

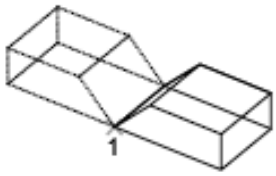
If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object are placed into the drawing and the original objects are retained.

**XY/YZ/ZX** Aligns the mirroring plane with one of the standard planes *(XY*, *YZ*, or *ZX)* through a specified point.

Specify point on (XY, YZ, ZX) plane <0,0,0>: *Specify a point (1) or press* ENTER




xy




yz


zx

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object are placed into the drawing and the original objects are retained.

**3 Points** Defines the mirroring plane by three points. If you select this option by specifying a point, the First Point on Mirror Plane prompt is not displayed.

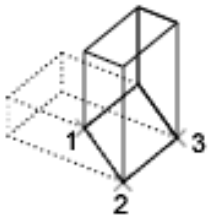
Specify first point on mirror plane: *Enter a point (1)*

Specify second point on mirror plane: *Enter a point (2)*

Specify third point on mirror plane: *Enter a point (3)*

Delete source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the reflected object is placed into the drawing and the original objects are deleted. If you enter **n** or press ENTER, the reflected object are placed into the drawing and the original objects are retained.




# MLEADER

## Quick Reference

Creates a line that connects annotation to a feature

Multileader 
Dimension ➤ MultileaderDoes not exist in the menus.
Multileaders panel, Multileader
**mleader**

Multileaders can be created arrowhead first (page 839), leader landing first (page 839), or content first (page 839). If a multileader style has been used, then the multileader can be created from that specified style.

Specify leader arrowhead location or [leader Landing first (page 839)/Content first (page 839)/Options (page 839)]

### Arrowhead First

Specifies a location for the arrowhead of the multileader object.

**Point Selection** Sets placement of the leader landing for the new multileader object.

Specify leader landing location:

If you exit the command at this point, then no text is associated with the multileader object.

### Leader Landing First

Specifies a location for the landing line of the multileader object.

If the multileader object you previously drew was landing first, then subsequent multileaders are created landing first until otherwise specified.

**Point Selection** Sets placement of the arrowhead for the new multileader object.

Specify leader arrowhead location:

If you exit the command at this point, then no text is associated with the multileader line.

### Content First

Specifies a location for the text or block associated with the multileader object.

If the multileader object you previously drew was content first, then subsequent multileader objects are created content first until otherwise specified.

**Point Selection** Sets placement for the text box for the text label associated with the multileader object. When you finish entering your text, click OK or click outside the text box.

You can also choose to place the multileader object with the leader first as outlined above.

If you choose End at this point, then no landing line is associated with the mleader object.

### Options

Specifies options for placing the multileader object.

Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options]:
**Leader Type** Specifies the type of leader line to use

Enter an option [Type/Landing]:

**Type** Specifies a straight, spline, or no leader line.

Select a leader type [Straight/sPline/None]:

**Landing** Changes the distance of the horizontal landing line

Use landing [Yes/No]:

If you choose No at this point, then no landing line is associated with the multileader object.

**Content Type** Specifies the type of content that will be used

Enter a content type [Block/None]:

**Block** Specifies a block within your drawing to associate with the new multileader.

Enter block name:

**None** Specifies no content type.

**Maxpoints** Specifies a maximum number of points for the new leader line.

Enter the maximum points for leader line or <none.:

**First Angle** Constrains the angle of the first point in the new leader line.

Enter first angle constraint or <none>:

**Second Angle** Constrains the second angle in the new leader line.

Enter second angle constraint or <none>:

**Exit Options** Returns you to the first MLEADER command prompt.

# MLEADERALIGN

## Quick Reference

Organizes selected multileaders along a specified line

Multileader 

Multileaders panel, Align Multileaders
mleaderalign
Select multileaders:

After multileaders have been selected, specify the multileader all others are aligned to.

Select multileader to align to or [Options]:

### Options

Specifies options for aligning the selected multileaders.

Enter an option [Distribute (page 841)/make leader segments Parallel (page 841)/specify Spacing (page 841)/Use current (page 841)]:
**Distribute** Spaces content evenly between two selected points.

**Make Leader Segments Parallel** Places content so that each of the last leader segments in the selected multileaders are parallel.

**Specify Spacing** Specifies spacing between the landing lines of the selected multileaders

**Use Current** Uses the current spacing between multileader content.

# MLEADERCOLLECT

## Quick Reference

Organizes selected multileaders containing blocks as content into a group attached to a single leader line

Multileader 
Multileaders panel, Collect Multileaders
mleadercollect
Select multileaders:

After you select multileaders, you can specify their placement in the drawing.

Specify collected multileader location or [Vertical/Horizontal/Wrap]:
**Specify Location** Specifies the point for placing the multileader collection in the upper-left corner of the collection.

**Vertical** Places the multileader collection in a vertical orientation.

**Horizontal** Places the multileader collection in a horizontal orientation.

**Wrap** Specifies a width for a wrapped multileader collection.

Specify wrap width or [Number]:

Number specifies a maximum number of blocks per row in the multileader collection.

# MLEADEREDIT

## Quick Reference

Adds leader lines to, or removes leader lines from, a multileader object

Multileader 
Multileaders panel, Add/Remove Multileader
**mleaderedit**
Select a multileader:
Select an option [Add leader/Remove leader]:
**Add Leader** Adds a leader line to the selected multileader object. The new leader line is added to the left or right of the selected multileader, depending on the location of the cursor.

Specify leader arrowhead location:

If there are more than 2 leader points in the specified multileader style, you are prompted to specify another point.

Specify next point:

**Remove Leader** Removes a leader line from the selected multileader object.

Specify leaders to remove:

# MLEADERSTYLE

## Quick Reference

Defines a new multileader style


Multileader Styles
Format ➤ Multileader StyleAt the Command prompt, enter mleaderstyle.
Multileaders panel, Multileader Style Manager
**mleaderstyle**

### Quick Reference

Defines a new multileader style

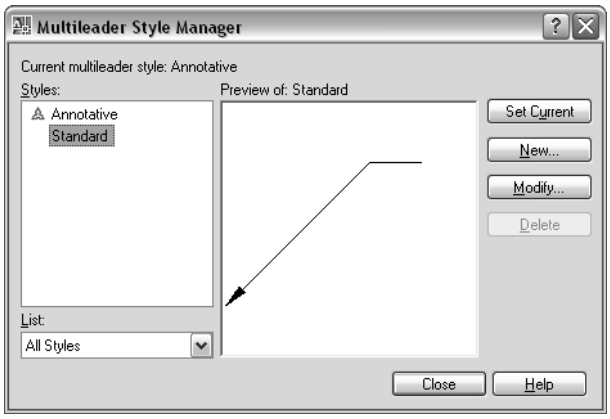
The Multileader Style Manager (page 843) is displayed.

## Multileader Style Manager

### Quick Reference


Multileader Styles
Format ➤ Multileader StyleAt the Command prompt, enter mleaderstyle.
Multileaders panel, Multileader Style Manager
**mleaderstyle**

Sets the current multileader style and creates, modifies, and deletes multileader styles.




### Current Multileader Style

Displays the name of the multileader style that is applied to multileaders you create. The default multileader style is STANDARD.

### Styles

Displays a list of multileader styles. The current style is highlighted.

### List

Controls the contents of the Styles list. Click All Styles to display all multileader styles available in the drawing. Click Styles In Use to display only the multileader styles that are referenced by multileaders in the current drawing.

### Preview

Displays a preview image of the style that is selected in the Styles list.

### Set Current

Sets the multileader style selected in the Styles list as the current style. All new multileaders are created using this multileader style.

### New

Displays the Create New Multileader Style dialog box, in which you can define new multileader styles.

#### Modify

Displays the Modify Multileader Style dialog box, in which you can modify multileader styles.

#### Delete

Deletes the multileader style selected in the Styles list. A style that is being used in the drawing cannot be deleted.

## Modify Multileader Style Dialog Box

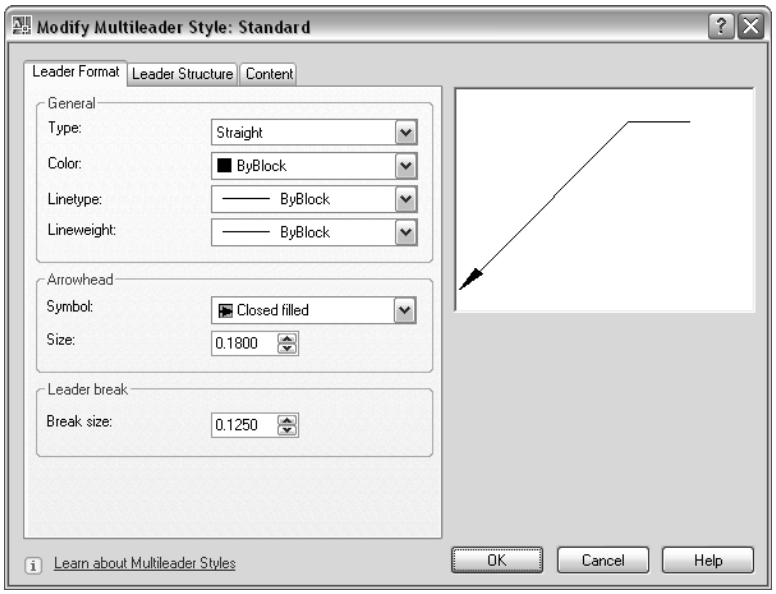
### Quick Reference


Multileader Styles

Format ➤ Multileader StyleAt the Command prompt, enter mleaderstyle.

**mleaderstyle**

Modifies an existing multileader style.




### Leader Format Tab

#### General

Controls the general appearance of the multileader.

**Type** Determines the type of leader line. You can choose a straight leader, a spline, or no leader line.

**Color** Determines the color of the leader line.

**Linetype** Determines the linetype of the leader line.

**Lineweight** Determines the lineweight of the leader line.

#### Arrowhead

Controls the appearance of the multileader arrowheads.

**Symbol** Sets the arrowhead symbol for the multileader.

**Size** Displays and sets the size of arrowheads.

#### Leader Break

Controls the settings used when adding a dimension break to a multileader.

**Break Size** Displays and sets the break size used for the DIMBREAK command when the multileader is selected.

### Leader Structure Tab

#### Constraints

Controls the constraints of the multileader.

**Maximum Leader Points** Specifies a maximum number of points for the leader line.

**First Segment Angle** Specifies the angle of the first point in the leader line.

**Second Segment Angle** Specifies the angle of the second point in the multileader landing line.

#### Landing Settings

Controls the landing settings of the multileader.

**Automatically Include Landing** Attaches a horizontal landing line to the multileader content.

**Set Landing Distance** Determines the fixed distance for the multileader landing line.

#### Scale

Controls the scaling of the multileader.

**Annotative** Specifies that the multileader is annotative. Click the information icon to learn more about annotative objects.

When the multileader is not annotative, the following options are available.

**Scale Multileaders to Layout** Determines a scaling factor for the multileader based on the scaling in the model space and paper space viewports.

**Specify Scale** Specifies the scale for the multileader.

### Content Tab

**Multileader Type** Determines whether the multileader contains text or a block.

When the multileader contains MTEXT, the following options are available.

### Text Options

Controls the appearance of the text for the multileader.

**Default Text** Sets default text for the multileader content. The [...] button launches the MTEXT In Place Editor.

**Text Style** Specifies a predefined text style for the attribute text. Currently loaded text styles are displayed. To load or create a text style, see the STYLE command (page 1367).

**Text Angle** Specifies the rotation angle of the multileader text.

**Text Color** Specifies the color of the multileader text.

**Text Height** Specifies the height of the multileader text.

**Always Left Justify** Specifies that the multileader text is always left justified.

**Frame Text Check Box** Frames the multileader text content with a text box.

### Leader Connection

Controls the leader connection settings of the multileader.

**Left Attachment** Controls the attachment of the landing line to the multileader text when the text is to the left of the leader.

**Right Attachment** Controls the attachment of the landing line to the multileader text when the text is to the right of the leader.

**Landing Gap** Specifies the distance between the landing line and the multileader text.

When the multileader contains a block, the following options are available.

### Block Options

Controls the properties of block content in a multileader object.

**Source block** Specifies the block used for multileader content.

**Attachment** Specifies the way the block is attached to the multileader object. You can attach the block by specifying the extents of the block, the insertion point of the block, or the center point of the block.

**Color** Specifies the color of the multileader block content. ByBlock is selected by default. The Block color control in the MLEADERSTYLE Content tab only takes effect if the object color included in the block is set to ByBlock.

### Preview

Displays a preview image of the style being modified.

### Learn About Multiline Styles

Click the link or information icon to learn more about multileaders and multileader styles.

# Create New Multileader Style Dialog Box

## Quick Reference

Multileader Styles

Format ➤ Multileader StyleAt the Command prompt, enter mleaderstyle.

**mleaderstyle**

Specifies a name for the new multileader style and specifies the existing multileader style on which the new multileader style will be based.


**New Style Name** Names the new multileader style.

**Start With** Specifies an existing multileader style whose settings are the default for the new multileader style.

**Annotative** Specifies that the multileader object is annotative. Click the information icon to learn more about annotative objects.

**Continue** Displays the Multileader Style Manager (page 843), in which you define the new multileader style.

# MLEDIT

## Quick Reference

Edits multiline intersections, breaks, and vertices

Modify ➤ Object ➤ MultilineAt the Command prompt, enter mledit.
**mledit**

The Multiline Edit Tools dialog box (page 850)is displayed.

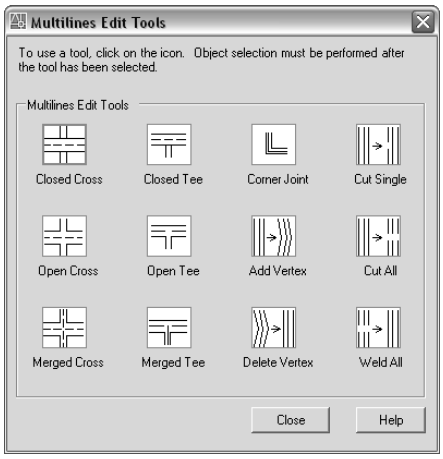
If you enter **-mledit** at the command prompt, options are displayed at the command prompt (page 858).

# Multiline Edit Tools Dialog Box

## Quick Reference

Modify ➤ Object ➤ MultilineAt the Command prompt, enter mledit.
**mledit**

Modifies multiline objects. The dialog box displays tools with sample images in four columns. The first column controls multilines that cross, the second controls multilines that form a *tee*, the third controls corner joints and vertices, and the fourth controls breaks in multilines.



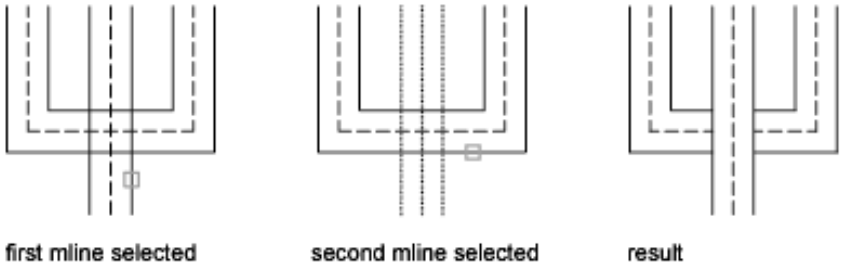
### Closed Cross

Creates a closed-cross intersection between two multilines.

Select first mline: *Select the foreground multiline*
Select second mline: *Select the intersecting multiline*

The closed-cross intersection is completed and the following prompt is displayed:

Select first mline or [Undo]: *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.




**Undo** Undoes the closed-cross intersection. The Select First Mline prompt is displayed.

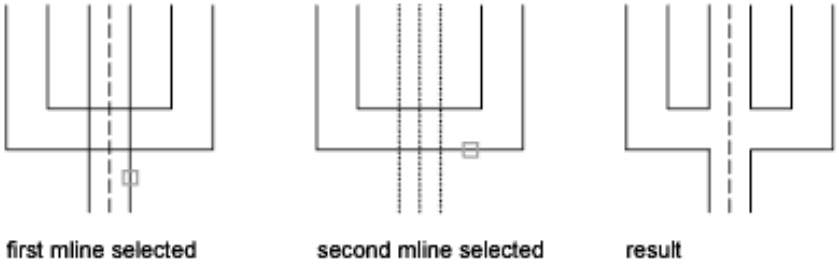
### Open Cross

Creates an open-cross intersection between two multilines. Breaks are inserted in all elements of the first multiline and only the outside elements of the second multiline.

Select first mline: *Select a multiline*
Select second mline: *Select the intersecting multiline*

The open-cross intersection is completed, and the following prompt is displayed:

Select first mline or [Undo]: *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.




**Undo** Undoes the open-cross intersection. The Select First Mline prompt is displayed.

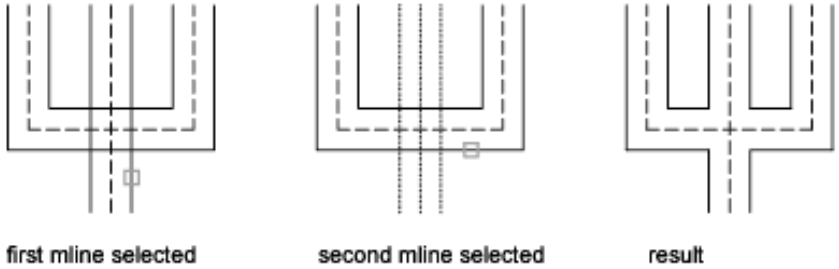
### Merged Cross

Creates a merged-cross intersection between two multilines. The order in which you select the multilines is not important.

Select first mline: *Select a multiline*
Select second mline: *Select the intersecting multiline*

The merged-cross intersection is completed, and the following prompt is displayed:

Select first mline or [Undo]: *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.



**Undo** Undoes the merged-cross intersection. The Select First Mline prompt is displayed.

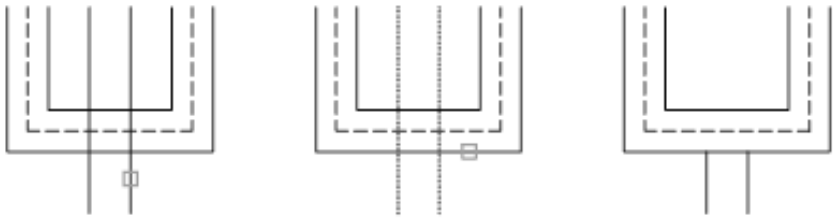
### Closed Tee

Creates a closed-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

Select first mline: *Select the multiline to trim*
Select second mline: *Select the intersecting multiline*

The closed-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.




**Undo** Undoes the closed-tee intersection. The Select First Mline prompt is displayed.

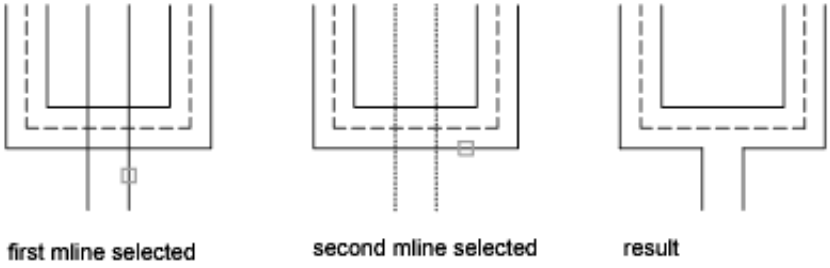
### Open Tee

Creates an open-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

Select first mline: *Select the multiline to trim or extend*
Select second mline: *Select the intersecting multiline*

The open-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.




**Undo** Undoes the open-tee intersection. The Select First Mline prompt is displayed.

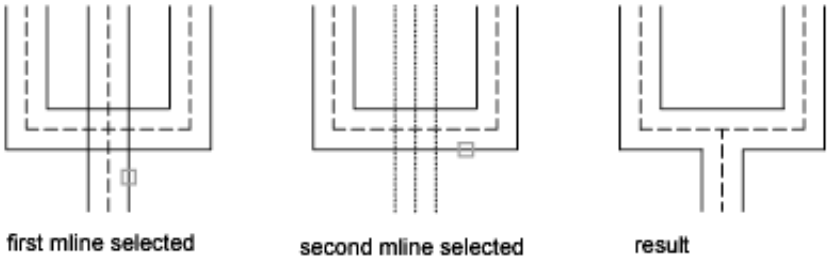
### Merged Tee

Creates a merged-tee intersection between two multilines.The multiline is trimmed or extended to its intersection with the other multiline.

Select first mline: *Select the multiline to trim or extend*
Select second mline: *Select the intersecting multiline*

The merged-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.



**Undo** Undoes the merged-tee intersection. The Select First Mline prompt is displayed.

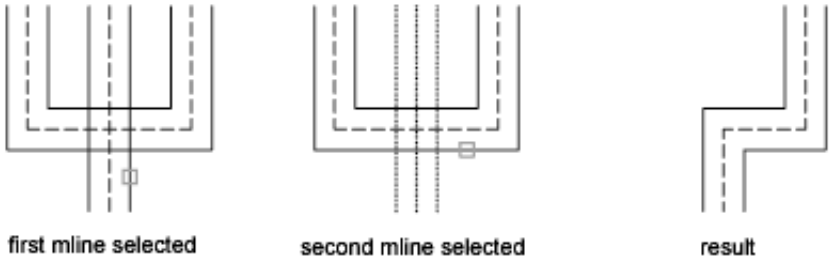
### Corner Joint

Creates a corner joint between multilines. The multilines are trimmed or extended to their intersection.

Select first mline: *Select the multiline to trim or extend*
Select second mline: *Select the second half of the corner*

The corner joint is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**
**First Mline** Edits another multiline. The Select Second Mline prompt is displayed.




**Undo** Undoes the corner joint. The Select First Mline prompt is displayed.

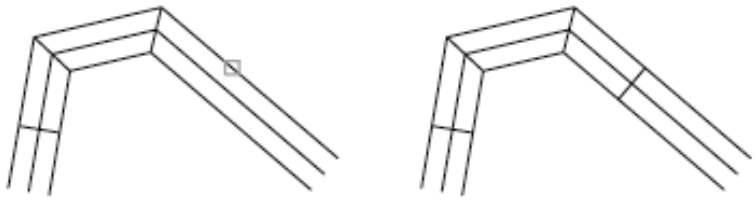
### Add Vertex

Adds a vertex to a multiline.

Select mline: *Select a multiline*

A vertex is added at the selected point, and the following prompt is displayed:

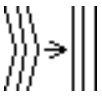
Select mline or [Undo]: *Select another multiline or enter* **u**
**Mline** Edits another multiline. The Select Mline prompt is displayed again.




**Undo** Removes the added vertex. The Select Mline prompt is displayed.

### Delete Vertex

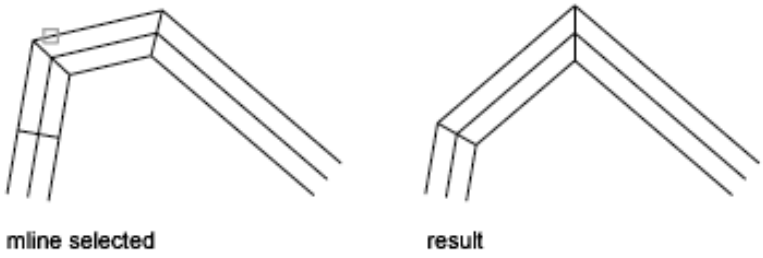
Deletes a vertex from a multiline.



Select mline: *Select a multiline*

The vertex nearest to the selected point is deleted, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**
**Mline** Edits another multiline. The Select Mline prompt is displayed again.




**Undo** Restores the deleted vertex. The Select Mline prompt is displayed.

### Cut Single

Creates a visual break in a selected element of a multiline.

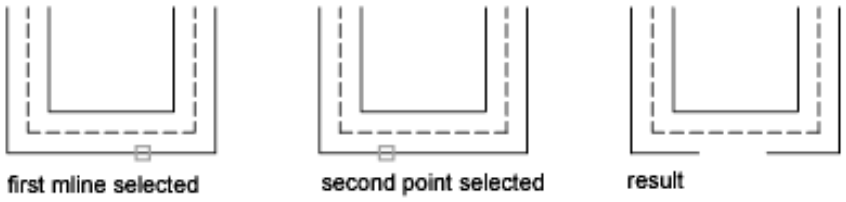
Select mline: *Select a multiline*

The selection point on the multiline is used as the first cut point, and the following prompt is displayed:

Select second point: *Specify the second cut point on the multiline*

The element is cut, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**
**Mline** Edits another multiline. The Select Mline prompt is displayed again.




**Undo** Undoes the cut. The Select Mline prompt is displayed.

### Cut All

Creates a visual break through the entire multiline.

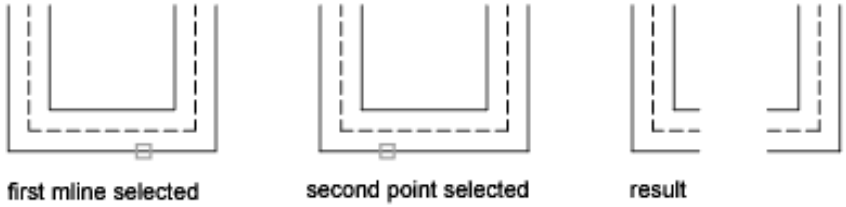
Select mline: *Select a multiline*

The selection point on the multiline is used as the first cut point, and the following prompt is displayed:

Select second point: *Specify the second cut point on the multiline*

All elements of the multiline are cut, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**
**Mline** Edits another multiline. The Select Mline prompt is displayed again.



**Undo** Undoes the cut. The Select Mline prompt is displayed.

#### Weld All

Rejoins multiline segments that have been cut.

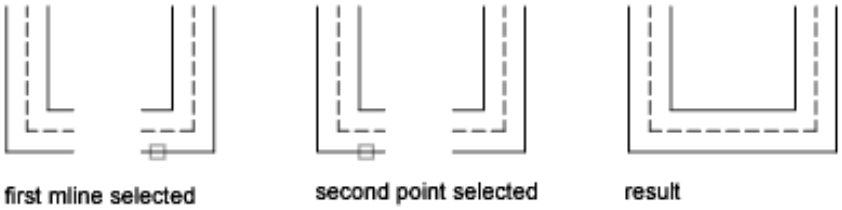
Select mline: *Select a multiline*

The selection point on the multiline is used as the start of the weld, and the following prompt is displayed:

Select second point: *Specify the end of the weld on the multiline*

The multiline is welded, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**
**Mline** Edits another multiline. The Select Mline prompt is displayed again.




**Undo** Undoes the weld. The Select Mline prompt is displayed again.

## -MLEDIT

### Quick Reference

If you enter **-mledit** at the command prompt, the following MLEDIT command prompts are displayed.

For more information about these command prompt options, see the Multiline Edit Tools dialog box. (page 850)

#### CC

Creates a closed-cross intersection between two multilines.

### OC

Creates an open-cross intersection between two multilines. Breaks are inserted in all elements of the first multiline and only the outside elements of the second multiline.

### MC

Creates a merged-cross intersection between two multilines. The order in which you select the multilines is not important.

### CT

Creates a closed-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

### OT

Creates an open-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

### MT

Creates a merged-tee intersection between two multilines.The multiline is trimmed or extended to its intersection with the other multiline.

### CJ

Creates a corner joint between multilines. The multilines are trimmed or extended to their intersection.

### AV

Adds a vertex to a multiline.

### DV

Deletes a vertex from a multiline.

### CS

Creates a visual break in a selected element of a multiline.

#### CA

Creates a visual break through the entire multiline.

#### WA

Rejoins multiline segments that have been cut.

# MLINE

## Quick Reference

Creates multiple parallel lines

Draw ➤ MultilineAt the Command prompt, enter mline.
**mline**
Current settings: Justification = *current*, Scale = *current*, Style = *current*
Specify start point (page 860) or [Justification (page 861)/Scale (page 861)/STyle (page 862)]: *Specify a point or enter an option*

#### Start Point

Specifies the next vertex of the multiline.

Specify next point:
Specify next point or [Undo]: *Specify a point or enter* **u**

If you create a multiline with two or more segments, the prompt includes the Close option.

Specify next point or [Close/Undo]: *Specify a point or enter an option*

**Next Point** Draws a multiline segment to the specified point using the current multiline style and continues to prompt for points.

**Undo** Undoes the last vertex point on the multiline. The previous prompt is displayed.

**Close** Closes the multiline by joining the last segments with the first segments.

### Justification

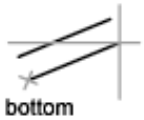
Determines how the multiline is drawn between the points you specify.

Enter justification type [Top/Zero/Bottom] <*current*>: *Enter an option or press* ENTER

**Top** Draws the multiline below the cursor, so that the line with the most positive offset is at the specified points.


**Zero** Draws the multiline with its origin centered at the cursor, so that the *MLSTYLE* Element Properties offset of 0.0 is at the specified points.


**Bottom** Draws the multiline above the cursor, so that the line with the most negative offset is at the specified points.




### Scale

Controls the overall width of the multiline. This scale does not affect linetype scale.

Enter mline scale <*current*>: *Enter a scale or press* ENTER

The scale factor is based on the width established in the multiline style definition. A scale factor of 2 produces a multiline twice as wide as the style definition. A negative scale factor flips the order of the offset line—the smallest on top when the multiline is drawn from left to right. A negative scale value also alters the scale by the absolute value. A scale factor of 0 collapses the multiline into a single line. For information about the multiline style definition, see *MLSTYLE*.


**Style**

Specifies a style to use for the multiline. See "Draw Multiline Objects" in the *User's Guide*, and see *MLSTYLE*.

Enter mline style name or [?]: *Enter a name or enter* **?**
**Style Name** Specifies the name of a style that has already been loaded or that's defined in a multiline library (MLN) file you've created.

**?—List Styles** Lists the loaded multiline styles.

# MLSTYLE

## Quick Reference

Creates, modifies, and manages multiline styles

Format ➤ Multiline StyleAt the Command prompt, enter mlstyle.
**mlstyle**

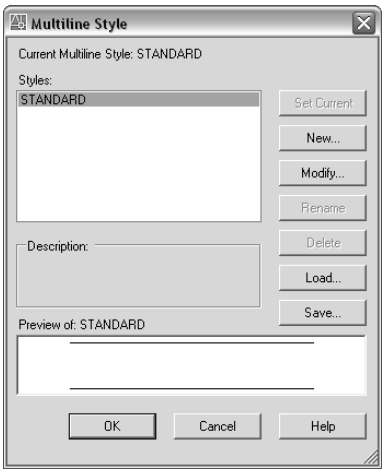
Displays the Multiline Style dialog box (page 862).

# Multiline Style Dialog Box

## Quick Reference

Format ➤ Multiline StyleAt the Command prompt, enter mlstyle.
**mlstyle**

Creates, modifies, saves, and loads multiline styles. The multiline style controls the number of elements and the properties of each element. MLSTYLE also controls the background color and the end caps of each multiline.




**Current Multiline Style** Displays the name of the current multiline style to be used for subsequently created multilines.

**Styles** Displays a list of multiline styles that are loaded in the drawing.

The list of multiline styles can include externally referenced multiline styles that is, multiline styles that exist in an externally referenced drawing (xref). Externally referenced multiline style names use the same syntax as other externally dependent nongraphical objects. See "Overview of Referenced Drawings (Xrefs)" in the *User's Guide*.

**Description** Displays the description of the selected multiline style.

**Preview Of** Displays the name and an image of the selected multiline style.

**Set Current** Sets the current multiline style for subsequently created multilines. Select a name from the Styles list and choose Set Current.

---

**NOTE** You cannot make a multiline style from an xref the current style.

---

**New** Displays the Create New Multiline Style dialog box (page 864), in which you can create a new multiline style.

**Modify** Displays the Modify Multiline Style dialog box (page 865), in which you can modify a selected multiline style. You cannot modify the default STANDARD multiline style.

**NOTE** You cannot edit the element and multiline properties of the STANDARD multiline style or any multiline style that is being used in the drawing. To edit an existing multiline style, you must do so before you draw any multilines that use the style.

**Rename** Renames the currently selected multiline style. You cannot rename the STANDARD multiline style.

**Delete** Removes the currently selected multiline style from the Styles list. It does not delete the style from the MLN file.

You cannot delete the STANDARD multiline style, the current multiline style, or a multiline style that is in use.

**Load** Displays the Load Multiline Styles dialog box (page 868), in which you can load multiline styles from a specified MLN file.

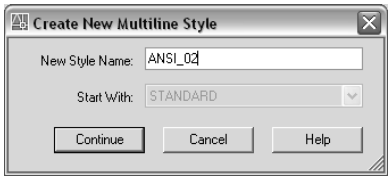
**Save** Saves or copies a multiline style to a multiline library (MLN) file. If you specify an MLN file that already exists, the new style definition is added to the file and existing definitions are not erased. The default file name is *acad.mln*.

# Create New Multiline Style Dialog Box

## Quick Reference

Format ➤ Multiline StyleAt the Command prompt, enter mlstyle.
**mlstyle**

Names the new multiline style and specifies the multiline style from which to start the new one.




**New Style Name** Names a new multiline style. The element and multiline properties are unavailable until you enter a new name and click Continue.

**Start With** Determines the multiline style from which to start the new one. To save time, choose a multiline style that is similar to the one that you want to create.

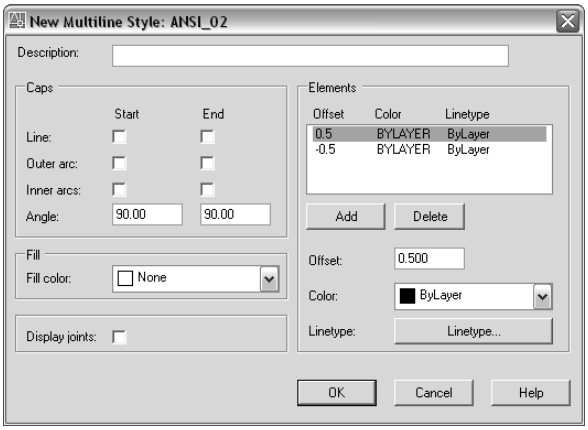
**Continue** Displays the New Multiline Style dialog box (page 865).

# New, Modify Multiline Style Dialog Boxes

## Quick Reference

Format ➤ Multiline Style
**mlstyle**

Sets the properties and elements for a new multiline style, or changes them for an existing multiline style.

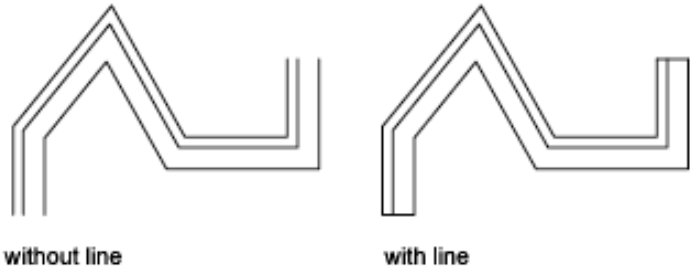



**Description** Adds a description to a multiline style. You can enter up to 255 characters, including spaces.

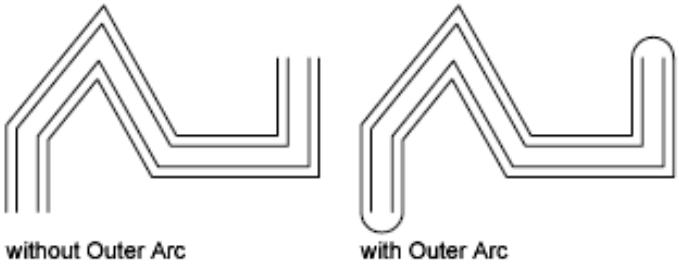
### Caps

Controls the start and end caps of the multiline.

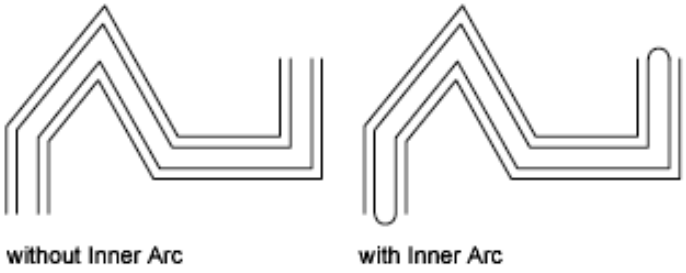
**Line** Displays a line segment across each end of the multiline.



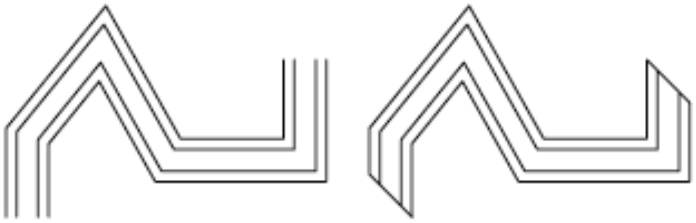

**Outer Arc** Displays an arc between the outermost elements of the multiline.




**Inner Arcs** Displays an arc between pairs of inner elements. If there's an odd number of elements, the center line is unconnected. For example, if there are six elements, inner arcs connect elements 2 and 5 and elements 3 and 4. If there are seven elements, inner arcs connect elements 2 and 6 and elements 3 and 5. Element 4 is left unconnected.




**Angle** Specifies the angle of the end caps.



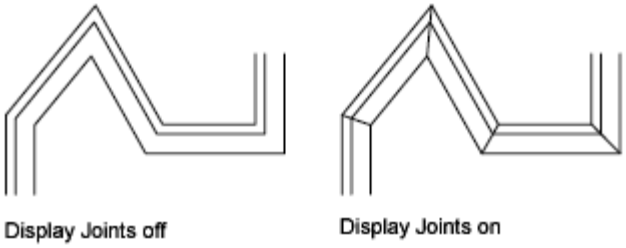

### Fill

Controls the background fill of the multiline.

**Fill Color** Sets the background fill color of the multiline. When you choose Select Color, the Select Color dialog box (page 251) is displayed.

### Display Joints

Controls the display of the joints at the vertices of each multiline segment. A joint is also known as a *miter*.




### Elements

Sets element properties, such as the offset, color, and linetype, of new and existing multiline elements.

**Offset Color Ltype** Displays all the elements in the current multiline style. Each element in the style is defined by its offset from the middle of the multiline, its color, and its linetype. Elements are always displayed in descending order of their offsets.

**Add** Adds a new element to the multiline style. Not available until color or linetype has been selected for a multiline style other than STANDARD.

**Delete** Deletes an element from the multiline style.

**Offset** Specifies the offset for each element in the multiline style.


a four-element multiline, each element offset from 0.0

**Color** Displays and sets the color for elements in the multiline style. When you choose Select Color, the Select Color dialog box (page 251) is displayed.

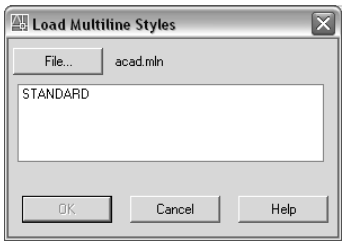
**Linetype** Displays and sets the linetype for elements in the multiline style. When you choose Linetype, the Select Linetype Properties dialog box is displayed, which lists loaded linetypes. To load a new linetype, click Load. The Load or Reload Linetypes dialog box (page 758) is displayed.

# Load Multiline Styles Dialog Box

## Quick Reference

Format ➤ Multiline StyleAt the Command prompt, enter mlstyle.
**mlstyle**

Loads a multiline style from an MLN file. The default file name is *acad.mln*. If *acad.mln* does not exist, or if it exists but cannot be found, click File to specify another file or file location.



**File** Displays a standard file selection dialog box (page 931) in which you can locate and select another multiline library file.

**List** Lists the multiline styles available in the current multiline library file. To load another multiline style, select a style from the list and click OK.

# MODEL

## Quick Reference

Switches from a layout tab to the Model tab

**model**

On the Model tab, you can create drawings in model space. The Model tab automatically sets the *TILEMODE* system variable to 1, and you can create model viewports to display various views of your drawing. Once you've completed your drawing, you can choose a layout tab to begin designing a layout environment from which to plot.

For possible performance gains when you switch between layout tabs or between the Model tab and a layout tab, use the *LAYOUTREGENCTL* system variable to control how the display list is updated.

# MOVE

## Quick Reference

Moves objects a specified distance in a specified direction

Modify 
Modify ➤ MoveDoes not exist in the menus.
Select the objects to move, and right-click in the drawing area. Click Move.
**move**
Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify base point or [Displacement]<Displacement>: *Specify a base point or enter* **d**
Specify second point or <use first point as displacement>: *Specify a point or press* ENTER

The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.

If you press ENTER at the Specify Second Point prompt, the first point is interpreted as a relative *X,Y,Z* displacement. For example, if you specify **2,3** for the base point and press ENTER at the next prompt, the objects move 2 units in the *X* direction and 3 units in the *Y* direction from their current position.

#### Displacement

Specify displacement <*last value*>: *Enter coordinates to represent a vector*

The coordinate values that you enter specify a relative distance and direction.

# MREDO

## Quick Reference

Reverses the effects of several previous UNDO or U commands

Standard 

**mredo**

Enter Number of Actions (page 870) number of actions or [All (page 870)/Last (page 870)]: *Specify an option, enter a positive number, or press* ENTER

**Number of Actions** Reverses the specified number of actions.

**All** Reverses all previous actions.

**Last** Reverses only the last action.

# MSLIDE

## Quick Reference

Creates a slide file of the current model viewport or the current layout

**mslide**

The Create Slide File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter a file name or select a slide (SLD) file from the list. A slide file is a raster image of a viewport.

In model space, MSLIDE makes a slide file of the current viewport only. In paper space, MSLIDE makes a slide file of the paper space display, including all viewports and their contents.

Off-screen portions of the drawing, or layers that are off or frozen, are not included in the slide.

Use the *VSLIDE* command to view slides, or the SLIDELIB (page 1893) utility to create a slide library.

When you view slides of images shaded with the SHADE command in a larger window or at a higher resolution than was used for creating the slide, black lines may appear interspersed among the lines of the shaded image. To avoid this situation, use a full screen that is set at the highest resolution when creating slides.

# MSPACE

## Quick Reference

Switches from paper space to a model space viewport

**mspace**

Commands operate in either model space or paper space. You use model space (the Model tab) to do drafting and design work and to create two-dimensional drawings or three-dimensional models. You use paper space (a layout tab) to create a finished layout of a drawing for plotting.

When you are in a layout, you can enter **mspace** at the command prompt to make the last viewport in the layout current, and then work in model space in that viewport within the layout. You can switch to model space by double-clicking a viewport, and you can switch to paper space by double-clicking an area of paper space.

# MTEDIT

## Quick Reference

Edits multiline text

**mtedit**
Select an MTEXT object:

The In-Place Text Editor (page 873)is displayed.

# MTEXT

## Quick Reference

Creates paragraphs of text as a single multiline text (mtext) object

Draw 

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.
Double-click a multiline text object.
**mtext**
Text panel, Multiline Text
Current text style: <*current*> Text height: <*current*> Annotative: <*current*>
Specify first corner:
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]:

After you specify the point for the opposite corner, the in-place text editor (page 873) is displayed.

If you specify one of the other options, or if you enter **-mtext** at the command prompt, MTEXT bypasses the In-Place Text editor and displays additional command prompts (page 888).

# In-Place Text Editor

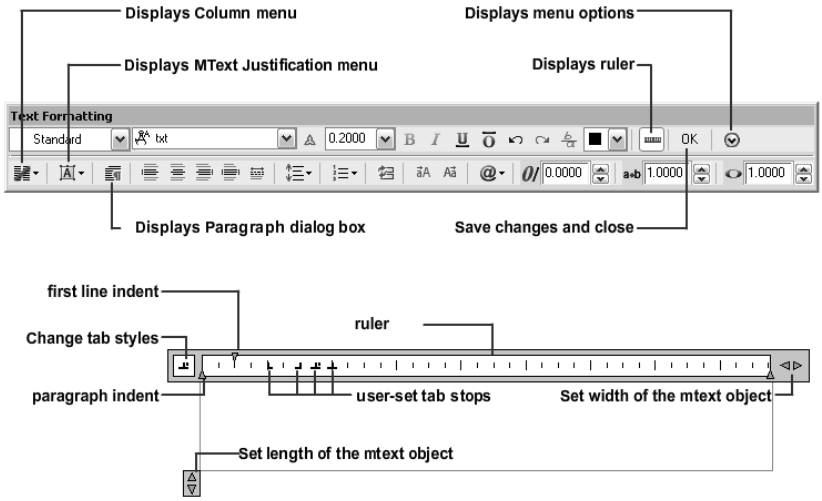
## Quick Reference

Draw 

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.

**mtext**

Creates or modifies single or multiline text objects. You can import or paste text from other files to use in multiline text, set tabs, adjust paragraph and line spacing and alignment, and create and modify columns.




The In-Place Text Editor includes a Text Formatting Toolbar (page 873), a Paragraph Dialog Box (page 880), a Columns Menu (page 881), and an Display Options Menu (page 879). When a table cell is selected for editing, the In-Place Text Editor displays column letters and row numbers.

### Text Formatting Toolbar

Controls the text style for a multiline text object and character and paragraph formatting for selected text.

**NOTE** Text that you paste from other word processing applications such as Microsoft Word will retain most of its formatting. With the options in Paste Special, you can strip out paragraph formatting such as paragraph-based alignment or character formatting from the pasted text.

**Style** Applies a text style to the multiline text object. The current style is saved in the *TEXTSTYLE* system variable.

Character formatting for font, height, and bold or italic attributes is overridden if you apply a new style to an existing multiline text object. Stacking, underlining, and color attributes are retained in characters to which a new style is applied.

Styles that have backward or upside-down effects are not applied. If a style defined with a vertical effect is applied to an SHX font, the text is displayed horizontally in the In-Place Text Editor.

**Font** Specifies a font for new text or changes the font of selected text. TrueType fonts are listed by font family name. AutoCAD compiled shape (SHX) fonts are listed by the name of the file in which the fonts are stored. Custom fonts and third-party fonts are displayed in the editor with Autodesk-supplied proxy fonts.

A sample drawing (*TrueType.dwg)* showing each font is provided in the *sample* directory.

**Annotative** Turns on or off for the current mtext object.

**Text Height** Sets the character height in drawing units for new text or changes the height of selected text. If the current text style has no fixed height, the text height is the value stored in the *TEXTSIZE* system variable. A multiline text object can contain characters of various heights.

**Bold** Turns bold formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

**Italic** Turns italic formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

**Underline** Turns underlining on and off for new or selected text.

**Overline** Turns overline on and off for new or selected text.

**Undo** Undoes actions in the In-Place Text Editor, including changes to either text content or text formatting. You can also use CTRL+Z.

**Redo** Redoes actions in the In-Place Text Editor, including changes to either text content or text formatting. You can also use CTRL+Y.

**Stack** Creates stacked text, for example, fractions, if the selected text contains stack characters. Also, unstacks text if stacked text is selected. When the stack characters, carat (^), forward slash (/), and pound sign (#), are used, the text to the left of the stack character is stacked on top of the text to the right.

By default, text that contains a carat converts to left-justified tolerance values. Text that contains the forward slash converts to center-justified fractional numbers; the slash is converted to a horizontal bar the length of the longer text string. Text that contains the pound sign converts to a fraction separated by a diagonal bar the height of the two text strings. The characters above the diagonal fraction bar are bottom-right aligned; the characters beneath the diagonal bar are top-left aligned.

**Text Color** Specifies a color for new text or changes the color of selected text.

You can assign text the color associated with the layer it is on (BYLAYER) or the color of the block it is contained in (BYBLOCK). You can also select one of the colors in the color list or click Other to open the Select Color dialog box (page 251).

**Ruler** Displays a ruler at the top of the editor. Drag the arrows at the end of the ruler to change the width of the mtext object. Also displays height and column grips when column mode is active.

You can also select tabs from the ruler. Clicking the Tab Selection button changes tab styles from left, center, right, and decimal. Once you make your selection, you adjust the corresponding tab in the ruler or in the Paragraph dialog box.

**OK** Closes the editor and saves any changes that you made.

**Options** Displays a list of additional text options. See Additional Text Options (page 878).

**Columns** Displays the column flyout menu, which provides three column options: No Columns, Static Columns, and Dynamic columns. See Columns menu (page 881).

**MText Justification** Displays the MText Justification menu with nine alignment options available. Top Left is the default.

**Paragraph** Displays the Paragraph dialog box. See the Paragraph dialog box (page 880) for a list of the options.

**Left, Center, Right, Justified and Distributed** Sets the justification and alignment for the left, center, or right text boundaries of the current or selected paragraph. Spaces entered at the end of a line are included and affect the justification of a line.

**Line Spacing** Displays suggested line spacing options or the Paragraph dialog box. Line spacing is set in the current or selected paragraph.

**NOTE** Line spacing is the distance between the bottom of the upper line and the top of the lower line of text in a multiple line paragraph.

The predefined options are:

- *1.0x*, *1.5x*, *2.0x*, or *2.5x*: Sets the line spacing at .5x increments in multiline text.
- *More:* Displays the Paragraph dialog box, which provides additional options.
- *Clear Paragraph Spacing:* Removes line spacing settings from the selected or current paragraph. The paragraph defaults to the mtext space setting.

Additional options in the Paragraph dialog box:

- *Exact:* Defines the space with an arbitrary unit value the user specifies. Changing text height will not affect line spacing.
- *Multiple:* Instead of assigning a value to line spacing, you specify spacing according to text height. When text height is not consistent in one line, the line space will be determined by the largest text height value in that line.
- *At least:* Takes both the user specified arbitrary value and the text height to determine spacing. If text height is smaller than the arbitrary value the line space is determined by the user specified value. If the text height is larger, the line spacing is equal to the text height value.

**NOTE** Not all of the new options for paragraph and paragraph line spacing are supported in AutoCAD 2007 and previous releases. See MTEXT Paragraph and Paragraph Line Spacing in Previous Releases in the *User's Guide*.

**Numbering** Displays the bullet and numbering menu.

Displays options for creating lists. (Not available for table cells.) The list is indented to align with the first selected paragraph.

- *Off:* When selected, removes letters, numbers, and bullets from selected text that has list formatting applied. Indentation is not changed.
- *Lettered:* Applies list formatting that uses letters with periods for the items in the list. If the list has more items than the alphabet has letters, the sequence continues by using double letters.

- *Numbered:* Applies list formatting that uses numbers with periods for the items in the list.
- *Bulleted:* Applies list formatting that uses bullets for the items in the list.
- *Restart:* Starts a new letter or number sequence in list formatting. If the selected items are in the middle of a list, unselected items below them also become part of the new list.
- *Continue:* Adds the selected paragraphs to the last list above and continues the sequence. If list items rather than paragraphs are selected, unselected items below the selected items continue the sequence.
- *Allow Auto-list:* Applies list formatting as you type. The following characters can be used as punctuation after letters and numbers and cannot be used as bullets: period (.), comma (,), close parenthesis ()), close angle bracket (>), close square bracket (]), and close curly bracket (}).
- *Use Tab Delimiter Only:* Limits the Allow Auto-list and Allow Bullets and Lists options. List formatting is applied to text only when the space after the letter, number, or bullet character was created by TAB, not SPACEBAR. This option is selected by default.
- *Allow Bullets and Lists:* When this option is selected, list formatting is applied to all plain text in the multiline text object that looks like a list. Text that meets the following criteria is considered to be a list. The line begins with (1) one or more letters or numbers or a symbol, followed by (2) punctuation after a letter or number, (3) a space created by pressing TAB, and (4) some text before the line is ended by ENTER or SHIFT+ENTER.
  When you clear the check mark, any list formatting in the multiline text object is removed and the items are converted to plain text. Allow Auto-list is turned off, and all the Bullets and Lists options are unavailable except Allow Bullets and Lists.

**Insert Field** Displays the Field dialog box (page 578), where you can select a field to insert in the text. When the dialog box closes, the current value of the field is displayed in the text.

**Uppercase** Changes the selected text to uppercase.

**Lowercase** Changes the selected text to lowercase.

**Symbol** Inserts a symbol or a nonbreaking space at the cursor position. You can also insert symbols manually. See Symbols and Special Characters (page 892).

Commonly used symbols are listed on the submenu, along with their control code or Unicode string. Click Other to display the Character Map dialog box, which contains the entire character set for each font available on your system. Select a character and click Select to place it in the Characters to Copy box. When you have selected all the characters that you want to use, click Copy to close the dialog box. In the editor, right-click and click Paste.

Symbols are not supported in vertical text.

**Oblique Angle** Determines the forward or backward slant of the text. The angle represents the offset from 90 degrees. Entering a value between -85 and 85 makes the text oblique. A positive obliquing angle slants text to the right. A negative obliquing angle slants text to the left.

**Tracking** Decreases or increases the space between the selected characters. The 1.0 setting is normal spacing. Set to more than 1.0 to increase spacing, and set to less than 1.0 to decrease spacing.

**Width Factor** Widens or narrows the selected characters. The 1.0 setting represents the normal width of the letter in this font. You can increase the width (for example, use a width factor of 2 to double the width) or decrease the width (for example, use a width factor of 0.5 for half the width).

### Additional Text Options

**Insert Field** Displays the Field dialog box.

**Symbol** Displays a list of available symbols. You can also select a Non-breaking space and open the Character Map for additional symbols.

**Import Text** Displays the Select File dialog box (a standard file selection dialog box (page 931)). Select any file that is in ASCII or RTF format. Imported text retains its original character formatting and style properties, but you can edit and format the imported text in the editor. After you select a text file to import, you can replace either selected text or all text, or append the inserted text to text selected within the text boundary. The file size for imported text is limited to 32 KB.

The editor automatically sets the text color to BYLAYER. When black characters are inserted and the background color is black, the editor automatically changes to white or the current color.

---

**NOTE** A Microsoft Excel spreadsheet imported into a drawing is truncated at 72 rows unless the spreadsheet was created in Microsoft Office 2002 with Service Pack 2 installed. The same limitation applies when the drawing that contains the OLE object is opened on a system with an earlier version of Microsoft Office installed; the spreadsheet is truncated.

---

**Paragraph Alignment** Sets alignment for the multiline text object. Top Left is the default setting. Spaces entered at the end of a line are included as part of the text and affect the justification of the line. Text is center-, left-, or right-justified with respect to the left and right text boundaries. Text is middle-, top-, or bottom-aligned with respect to the top and bottom text boundaries. See MTEXT command line (page 888) for an illustration of the nine justification options.

**Paragraph** Displays options for paragraph formatting. See the Paragraph dialog box (page 880).

**Bullets and Lists** Displays the options for numbering lists.

**Columns** Displays options for columns. See the Columns menu (page 881).

**Find and Replace** Displays the Find and Replace dialog box (page 884).

**Change Case** Changes the case of selected text. Options are Uppercase and Lowercase.

**AutoCAPS** Converts all new and imported text to uppercase. AutoCAPS does not affect existing text. To change the case of existing text, select the text and right-click. Click Change Case.

**Character Set** Displays a menu of code pages. Select a code page to apply it to the selected text.

**Combine Paragraphs** Combines selected paragraphs into a single paragraph and replaces each paragraph return with a space.

**Remove Formatting** Removes character formatting for selected characters, paragraph formatting for a selected paragraph, or all formatting from a selected paragraph.

**Background Mask** Displays the Background Mask dialog box (page 883). (Not available for table cells.)

**Editor Settings** Displays a list of options for the Text Formatting toolbar. See the Display Options menu (page 879).

**Learn About MText** Displays the New Features Workshop, which contains an overview of the MText feature.

### Display Options Menu

Changes the behavior of the Text Formatting toolbar and provides additional editing options. Options are specific to the Display Options menu and are not available on the Text Formatting toolbar.

**NOTE** Some options may not be available depending on what you are editing.

**Show Toolbar** Controls display of the Text Formatting toolbar. To restore display of the toolbar, right-click in the editor and click Option ➤ Show Toolbar.

**Show Options** Expands the Text Formatting toolbar to show more options.

**Show Rulers** Controls the display of the ruler.

**Opaque Background** When selected, makes the background of the editor opaque. By default, the editor is transparent. (Not available for table cells.)

**Text Highlight Color** Displays the AutoCAD generic Select Color dialog box. Specifies the highlight color when text is selected.

# Paragraph Dialog Box

## Quick Reference

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.
**mtext**

Sets indentation for paragraphs and the first lines of paragraphs. Specifies tab stops, indents, controls paragraph alignment, paragraph spacing, and paragraph line spacing.

**Tab** Displays tab setting options, including adding and removing tabs. Options include setting left, center, right, and decimal tabs. You can also set tabs from the Tab selection button on the In-Place Editor's ruler.

**Decimal style** Sets the decimal style based on current user locale. Options include setting the decimal style as period, comma, and space. This setting is preserved with the drawing even if the locale setting is changed.

**Left Indent** Sets the indent value for the first line or hanging indent to the selected or current paragraphs.

**Right Indent** Applies the indent to the entire selected or current paragraph.

**Paragraph Alignment** Sets the alignment properties for the current or selected paragraphs.

**Paragraph Spacing** Specifies the spacing before or after the current or selected paragraphs. The distance between two paragraphs is determined by the total

of the after paragraph spacing value of the upper paragraph and the before paragraph spacing value of the lower paragraph.

**Paragraph Line Spacing** Sets the spacing between individual lines in the current or selected paragraphs.

---

**NOTE** Not all of the new options for paragraph and paragraph line spacing are supported in AutoCAD 2007 and previous releases. See MTEXT Paragraph and Paragraph Line Spacing in Previous Releases in the *User's Guide*.

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Draw

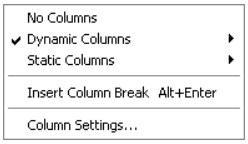
# Columns Menu

## Quick Reference

Draw ➤ Text ➤ Multiline Text
**mtext**

Allows you to format an mtext object into multiple columns. You can specify column and gutter width, height, and number of columns. You can edit column width and height with grip-editing.

To create multiple columns you always start with a single column. Depending on the column mode you choose, you have two different methods for creating and manipulating columns – static mode or dynamic mode.




The following options are available from the Columns menu:

**No Columns** Specifies no columns for the current mtext object.

**Dynamic Columns** Sets dynamic columns mode to the current mtext object. Dynamic columns are text driven. Adjusting columns affects text flow and text flow causes columns to be added or removed. Auto height or Manual height options are available.

**Static Columns** Sets static columns mode to the current mtext object. You can specify the total width and height of the mtext object, and the number

of columns. All the columns share the same height and are aligned at both sides.

**Insert Column Break Alt+Enter** Inserts a manual column break. This is disabled when you select No Columns.

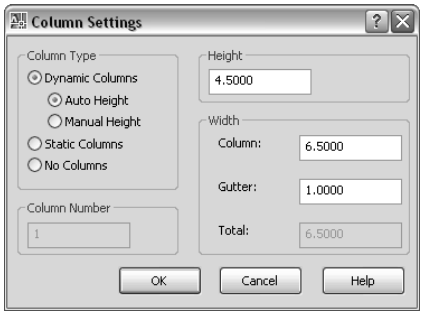
**Column Settings** Displays Column Settings dialog box.

# Column Settings Dialog Box

## Quick Reference

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.
**mtext**

Displays options for setting up columns such as the type, number of columns, height, and width, and gutter size.




**Column Type** Displays choices for the type of columns you want to create.

**Column Number** Sets the number of columns. This is only active when you select Static Columns.

**Height** Displays mtext height when Auto Height with Dynamic or Static Columns is selected.

**Width** Displays and specifies control column and gutter width values. The gutter value is five times the default mtext text height.

Also displays the total width value of the mtext object. When Dynamic Columns is selected, this is read-only.

Draw

# Background Mask Dialog Box

## Quick Reference

Draw

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.

**mtext**

Controls using an opaque background behind multiline text.

### Use Background Mask

Puts an opaque background behind the text.

**NOTE** When you apply a background mask to mtext multiple columns only the column areas will be masked. The space between the columns commonly referred to as gutters will be unmasked.

### Border Offset Factor

Specifies the margin around the text for the opaque background. The value is based on the text height. A factor of 1.0 exactly fits the multiline text object. A factor of 1.5 extends the background by 0.5 times the text height.

### Fill Color

Specifies the color for the background.

**Use Background Drawing Color** Provides a background that is the same color as the background of the drawing.

**Color** Specifies a color for the opaque background. You can select one of the colors in the list or click Select Color to open the Select Color dialog box (page 251).

# Find and Replace Dialog Box

## Quick Reference

Draw 

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.

**mtext**

Searches for specified text strings and replaces them with new text.

**Find What** Provides a space for you to type the text you want to find.

**Replace With** Provides a space for you to type the text you want to replace the text you typed in Find What.

**Find Next** Finds the next instance of the text specified in Find What.

**Replace** Finds the next instance of the text specified in Find What and replaces it with the text in Replace With.

**Replace All** Finds all instances of the text specified in Find What and replaces it with the text in Replace With.

**Match Whole Word Only** Finds only whole words specified in Find What.

Text that is part of another word is ignored. When this option is cleared, a match is found for text strings, whether they are single words or parts of other words.

**Match Case** Finds only text with uppercase and lowercase characters as specified in Find What.

The case of all characters in the text string must match the case of the text in Find What. When this option is cleared, a match is found for specified text strings regardless of case.

# Stack Properties Dialog Box

## Quick Reference

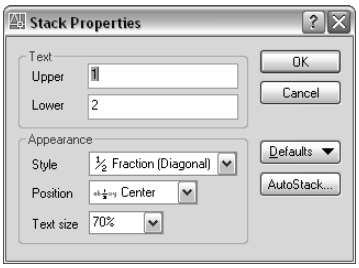
Draw

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.

**mtext**

Edits the text, stack type, alignment, and size of stacked text. To open the Stack Properties dialog box, select the stacked text, right-click, and click Properties on the shortcut menu.

You can edit the upper and lower text separately. The Appearance options control the stack style, position, and text size of the stacked text.




### Text

Changes the upper and lower numbers of a stacked fraction.

**Upper** Edits the number in the upper part or first half of a stacked fraction.

**Lower** Edits the number in the lower part or second half of a stacked fraction.

### Appearance

Edits the style, position, or text size of a stacked fraction.

### Style

Specifies a style format for stacked text: horizontal fraction, diagonal fraction, tolerance, and decimal.

**Fraction (Horizontal)** Stacks the selected text with the first number on top of the second number separated by a horizontal line.

**Fraction (Diagonal)** Stacks the selected text with the first number on top of the second number separated by a diagonal line.

---

**NOTE** Releases of AutoCAD earlier than AutoCAD 2000 do not support diagonal fractions. If a multiline text object contains diagonal fractions, the fractions are converted to horizontal fractions when you save the drawing to pre-AutoCAD 2000 releases. Diagonal fractions are restored when the drawing is re-opened in AutoCAD 2000 or a later release. If a single multiline text object contains both horizontal and diagonal fractions, all fractions are converted to diagonal fractions when the drawing is reopened in AutoCAD 2000 or a later release.

---

**Tolerance** Stacks the selected text with the first number on top of the second number. There is no line between the numbers.

**Decimal** A variation of the Tolerance style that aligns the decimal point of both the upper and lower numbers of selected text.

### Position

Specifies how fractions are aligned. Center alignment is the default. All stacked text in an object uses the same alignment.

**Top** Aligns the top of the fraction with the top of the text line.

**Center** Centers the fraction vertically at the center of the text line.

**Bottom** Aligns the bottom of the fraction with the text baseline.

### Text Size

Controls the size of the stacked text as a percentage of the size of the current text style (from 25 to 125 percent). Default text size is 70 percent.

### Defaults

Saves the new settings as defaults or restores the previous default values to the current stacked text.

### AutoStack Button

Displays the AutoStack Properties dialog box (page 887). AutoStack only stacks numeric characters immediately before and after the carat, slash, and pound

characters. To stack nonnumeric characters, or text that includes spaces, select the text and choose the Stack button.

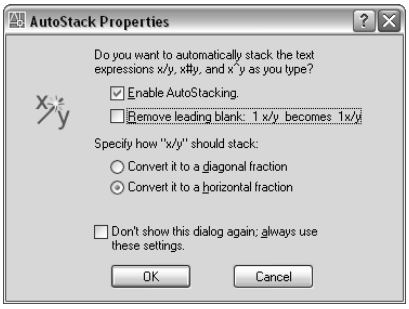
# AutoStack Properties Dialog Box

## Quick Reference

Draw 

Draw ➤ Text ➤ Multiline TextAt the Command prompt, enter mtext.
**mtext**

Sets the defaults for automatically stacking characters.




**Enable AutoStacking** Automatically stacks numeric characters entered before and after the carat, slash, or pound character. For example, if you type **1#3** followed by a nonnumeric character or space, the text is automatically stacked as a diagonal fraction.

**Remove Leading Blank** Removes blanks between a whole number and a fraction. This option is available only when AutoStacking is turned on.

**Convert It to a Diagonal Fraction** Converts the slash character to a diagonal fraction when AutoStack is on.

**Convert It to a Horizontal Fraction** Converts the slash character to a horizontal fraction when AutoStack is on.

---

**NOTE** Whether AutoStack is on or off, the pound character is always converted to a diagonal fraction, and the carat character is always converted to a tolerance format.

---

**Don't Show This Dialog Again; Always Use These Settings** Suppresses display of the AutoStack Properties dialog box. The current property settings are used for all stacked text. When this option is cleared, the AutoStack Properties dialog box is automatically displayed if you type two numbers separated by a slash, carat, or pound sign followed by a space or nonnumeric character.

**NOTE** To display the AutoStack Properties dialog box when you have suppressed automatic display, select stacked text, right-click, and click Properties on the shortcut menu. In the Stack Properties dialog box, click AutoStack.

# -MTEXT

## Quick Reference

If you enter **-mtext** at the command prompt, MTEXT bypasses the In-Place Text Editor and displays additional command prompts.

Current text style: <*current*> Text height: <*current*> Annotative: <*current*>
Specify first corner:
Specify opposite corner (page 888) or [Height (page 888)/Justify (page 889)/Line spacing (page 890)/Rotation (page 891)/Style (page 891)/Width (page 892)/Columns (page 892)]:

### Opposite Corner

As you drag the pointing device to specify the opposite corner, a rectangle is displayed to show the location and size of the multiline text object. Arrows within the rectangle indicate the direction of the paragraph's text flow.

### Height

Specifies the text height to use for multiline text characters.

Specify height <*current*>: *Specify a point (1), enter a value, or press* ENTER

The Specify Height prompt is displayed only if the current text style is not .

Specify paper text height <*current*>: *Specify a height, or press* ENTER

The Specify Paper Text Height prompt is displayed only if the current text style is annotative.

The default height, if nonzero, is the height of the current style; otherwise, it is the height stored in the *TEXTSIZE* system variable. Character height is

calculated in drawing units. Changing the height updates the value stored in *TEXTSIZE*.

### Justify

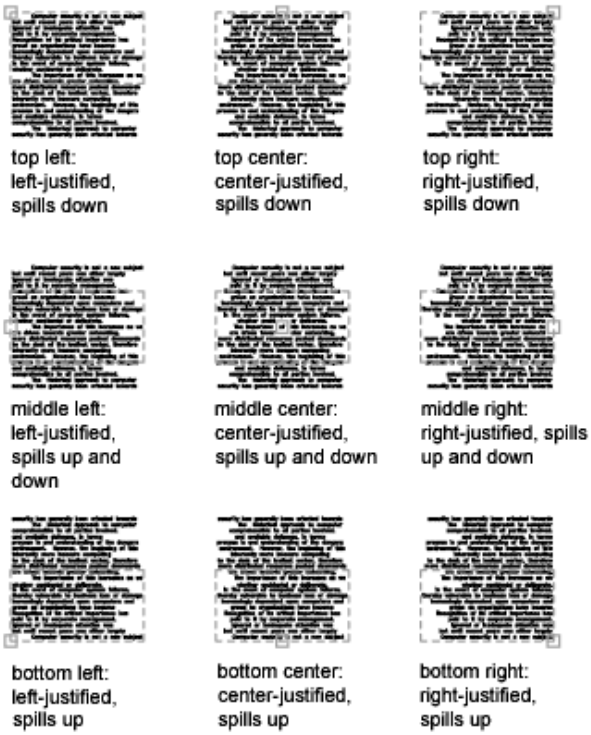
Determines both text justification and text flow, for new or selected text, in relation to the text boundary. The current justification (Top Left, by default) is applied to new text. The text is justified within the specified rectangle based on the justification setting and one of nine justification points on the rectangle. The justification point is based on the first point used to specify the rectangle. Text is center-, left-, or right-justified with respect to the left and right text boundaries. Spaces entered at the end of a line are included as part of the text and affect the justification of the line. Text flow controls whether text is aligned from the middle, the top, or the bottom of the paragraph with respect to the top and bottom text boundaries.

Enter justification [TL/TC/TR/ML/MC/MR/BL/BC/BR] <TL>:

**Justify options**

| Option | Meaning |
|---|---|
| TL | Top Left |
| TC | Top Center |
| TR | Top Right |
| ML | Middle Left |
| MC | Middle Center |
| MR | Middle Right |
| BL | Bottom Left |
| BC | Bottom Center |
| BR | Bottom Right |

The following illustrations show each justification option.




### Line Spacing

Specifies line spacing for the multiline text object. Line spacing is the vertical distance between the bottom (or baseline) of one line of text and the bottom of the next line of text.

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**NOTE** Exact spacing is recommended when you use MTEXT to create a table. Use a smaller text height than the specified line spacing to ensure that text does not overlap.

---

Enter line spacing type [At least/Exactly] <*current*>:
**At Least** Adjusts lines of text automatically based on the height of the largest character in the line. When At Least is selected, lines of text with taller characters have added space between lines.

Enter line spacing factor or distance <*current*>:

- *Spacing Factor:* Sets the line spacing to a multiple of single-line spacing. Single spacing is 1.66 times the height of the text characters. You enter the spacing increment as a number followed by **x** to indicate a multiple

of single spacing. For example, specify single spacing by entering **1x**, or specify double spacing by entering **2x**.

- *Distance:* Sets the line spacing to an absolute value measured in drawing units. Valid values must be between 0.0833 (0.25x) and 1.3333 (4x).

**Exactly** Forces the line spacing to be the same for all lines of text in the multiline text object. Spacing is based on the text height of the object or text style.

Enter line spacing factor or distance <*current*>:

- *Spacing Factor:* Sets the line spacing to a multiple of single-line spacing. Single spacing is 1.66 times the height of the text characters. You can enter a spacing factor as a number followed by **x** to indicate a multiple of single spacing. For example, specify single spacing by entering **1x**, or specify double spacing by entering **2x**.
- *Distance:* Sets the line spacing to an absolute value measured in drawing units. Valid values must be between 0.0833 (0.25x) and 1.3333 (4x).

### Rotation

Specifies the rotation angle of the text boundary.

Specify rotation angle <*current*>: *Specify a point or enter a value*

If you use the pointing device to specify a point, the rotation angle is determined by the angle between the *X* axis and the line defined by the most recently entered point (default 0,0,0) and the specified point.

The previous prompt is redisplayed until you specify the opposite corner of the text boundary.

### Style

Specifies the text style to use for multiline text.

Enter style name or [?] <*current*>:
**Style Name** Specifies a text style name. Text styles can be defined and saved using the *STYLE* command.

**?—List Styles** Lists text style names and characteristics.

The previous prompt is redisplayed until you specify the opposite corner of the text boundary.

### Width

Specifies the width of the text boundary.

Specify width: *Specify a point or enter a value*

If you use the pointing device to specify a point, the width is calculated as the distance between the start point and the specified point. Words within each line of the multiline text object wrap to fit the width of the text boundary. If you specify a width of 0, word wrap is turned off and the width of the multiline text object is as wide as the longest line of text. You can end a line of text at a specific point by typing the text and pressing ENTER. To end the command, press ENTER at the MTEXT prompt.

### Columns

Specifies the column options for an mtext object.

Enter column type: *Specify a column option*
**Static** Specifies the total column width, the number of columns, the gutter width (the space between the columns), and the height of columns.

**Dynamic** Specifies column width, gutter width and column height. Dynamic columns are text driven. Adjusting columns affect text flow and text flow causes columns to be added or removed.

**No columns** Sets no column mode to current mtext object.

# Symbols and Special Characters

## Quick Reference

You can enter the following special characters and symbols by entering a control code or a Unicode string. Or in the In-Place Text Editor, click Symbol on the expanded toolbar.

**NOTE** Symbols are not supported in vertical text.

**Unicode strings and control codes**

| Control Codes | Unicode Strings | Result |
|---|---|---|
| **%%d** | **\U+00B0** | Degree symbol (°) |

**Unicode strings and control codes**

| Control Codes | Unicode Strings | Result |
|---|---|---|
| %%p | \U+00B1 | Tolerance symbol (±) |
| %%c | \U+2205 | Diameter symbol (  ) |

To insert the following text symbols, click Symbol on the expanded Text Formatting toolbar or enter the appropriate Unicode string:

**Text symbols and Unicode strings**

| Name | Symbol | Unicode String |
|---|---|---|
| Almost equal | ≈ | \U+2248 |
| Angle | ∠ | \U+2220 |
| Boundary line |  | \U+E100 |
| Centerline |  | \U+2104 |
| Delta | Δ | \U+0394 |
| Electrical phase | ϕ | \U+0278 |
| Flow line |  | \U+E101 |
| Identity | ≡ | \U+2261 |

**Text symbols and Unicode strings**

| Name | Symbol | Unicode String |
|---|---|---|
| Initial length | ℓ | **\U+E200** |
| Monument line | ℳ | **\U+E102** |
| Not equal | ≠ | **\U+2260** |
| Ohm | Ω | **\U+2126** |
| Omega | Ω | **\U+03A9** |
| Plate/property line | ℙ | **\U+214A** |
| Subscript 2 | ₂ | **\U+2082** |
| Squared | ² | **\U+00B2** |
| Cubed | ³ | **\U+00B3** |

These text symbols are available in the following True Type (TTF) and SHX fonts:

- Simplex
- RomanS
- Isocp
- Isocp2

- Isocp3
- Isoct
- Isoct2
- Isoct3
- Isocpeur (TTF only)
- Isocpeur italic (TTF only)
- Isocteur (TTF only)
- Isocteur italic (TTF only)

# MULTIPLE

## Quick Reference

Repeats the next command until canceled

**multiple**
Enter command name to repeat:

The command that you enter is repeated until you press ESC. Because MULTIPLE repeats only the command name, any parameters must be specified each time.

MULTIPLE does not repeat commands that display dialog boxes.

**NOTE** You cannot use MULTIPLE as an argument to the AutoLISP® command function.

# MVIEW

## Quick Reference

Creates and controls layout viewports

View ➤ Viewports ➤ 1 Viewport, 2 Viewports, 3 Viewports, 4 Viewports
**mview**

Specify corner of viewport (page 896) or [ON (page 896)/OFF (page 896)/Fit (page 896)/Shadeplot (page 897)/Lock (page 897)/Object (page 897)/Polygonal (page 897)/Restore (page 898)/LAyer (page 898)/2 (page 898)/3 (page 899)/4 (page 899)] <Fit>: *Enter an option or specify a point*

In a layout, you can create as many viewports as you want, but only up to 64 viewports can be active at one time (see *MAXACTVP*). Objects in model space are visible only in active viewports. Viewports that are not active are blank. Use the On and Off options to control whether viewports are active.

### Corner of Viewport

Specifies the first corner of a rectangular viewport.

Specify opposite corner:

### On

Makes a selected viewport active. An active viewport displays objects in model space. The MAXACTVP system variable controls the maximum number of viewports that can be active at one time. If your drawing contains more viewports than the number specified in MAXACTVP, you must turn one off to make another one active.

Select objects: *Select one or more viewports*

### Off

Makes a selected viewport inactive. Objects in model space are not displayed in an inactive viewport.

Select objects: *Select one or more viewports*

### Fit

Creates one viewport that fills the layout to the edges of the printable area. When the paper background and printable area are turned off, the viewport fills the display.

fit

### Shadeplot

Specifies how viewports in layouts are plotted.

Shade plot? [As displayed/*all visual styles*/*all render presets*] <*As displayed*>: *Enter a shade plot option*
**As Displayed** Specifies that a viewport is plotted the same way it is displayed.

**All Visual Styles** Specifies that a viewport is plotted using the specified visual style. All visual styles in the drawing are listed as options whether in use or not.

**All Render Presets** Specifies that a viewport is plotted using the specified render preset. All render presets are listed as options.

Select objects: *Select one or more viewports*

### Lock

Prevents the zoom scale factor in the selected viewport from being changed when working in model space.

Viewport View Locking [ON/OFF]: *Enter* **on** *or* **off**
Select objects: *Select one or more viewports*

### Object

Specifies a closed polyline, ellipse, spline, region, or circle to convert into a viewport. The polyline you specify must be closed and contain at least three vertices. It can be self-intersecting, and it can contain an arc as well as line segments.

Select object to clip viewport: *Select an object*

### Polygonal

Creates an irregularly shaped viewport using specified points. The prompts are similar to those displayed when you specify a polygonal clip boundary for external references (xrefs), but you can specify arcs when you create a polygonal viewport boundary.

Specify start point: *Specify a point*
Specify next point or [Arc/Close/Length/Undo]: *Specify a point or enter an option*
**Arc** Adds arc segments to the polygonal viewport.

[Angle/CEnter/CLose/Direction/Line/Radius/Second pt/Undo/Endpoint of arc] <Endpoint>: *Enter an option or press* ENTER

For a description of the options for creating arc segments, see the Arc option in *PLINE*.

**Close** Closes the boundary. If you press ENTER after specifying at least three points, the boundary is closed automatically.

**Length** Draws a line segment of a specified length at the same angle as the previous segment. If the previous segment is an arc, the new line segment is drawn tangent to that arc segment.

**Undo** Removes the most recent line or arc segment added to the polygonal viewport.

### Restore

Restores viewport configurations saved with the *VPORTS* command.

Enter viewport configuration name or [?] <*ACTIVE>: *Enter* **?**, *enter a name, or press* ENTER
Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER
**First Corner** Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

**Fit** Sizes the viewports to fill the drawing area.

### Layer

Resets layer property overrides for the selected viewport to their global layer properties.

Reset viewport layer property overrides back to global properties [Y/N]?: *Enter* **Y** *to remove all layer property overrides*
Select viewports: *Select a single or multiple viewports and press* ENTER

### 2

Divides the specified area horizontally or vertically into two viewports of equal size.

Enter viewport arrangement [Horizontal/Vertical] <Vertical>: *Enter* **h** *or press* ENTER
Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER

2/vertical

**First Corner** Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

**Fit** Sizes the viewports to fill the drawing area.

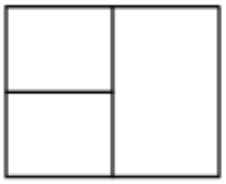
**3**

Divides the specified area into three viewports.

Enter viewport arrangement
[Horizontal/Vertical/Above/Below/Left/<Right>: *Enter an option or press* ENTER

The Horizontal and Vertical options split the specified area into thirds. The other options split the area into three viewports: one large viewport and two smaller ones. The Above, Below, Left, and Right options specify where the larger viewport is placed.

Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER



3/right

**First Corner** Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

**Fit** Sizes the viewports to fill the drawing area.

**4**

Divides the specified area horizontally and vertically into four viewports of equal size.

Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER

4

**First Corner** Positions and sizes new viewports using the window selection method; he viewports are fit into the selected area.

**Fit** Sizes the viewports to fill the drawing area.

# MVSETUP

## Quick Reference

Sets up the specifications of a drawing

**mvsetup**

When you enter **mvsetup,** the command prompts displayed depend on whether you are on the Model tab (model space) (page 901) or on a layout tab (paper space) (page 901).

On the Model tab, you set the units type, drawing scale factor, and paper size at the command prompt using MVSETUP. Using the settings you provide, a rectangular border is drawn at the grid limits.

On a layout tab, you can insert one of several predefined title blocks into the drawing and create a set of layout viewports within the title block. You can specify a global scale as the ratio between the scale of the title block in the layout and the drawing on the Model tab. The Model tab is most useful for plotting multiple views of a drawing within a single border.

To easily specify all layout page settings and prepare your drawing for plotting, you can also use the Page Setup dialog box, which is automatically displayed when you select a layout in a new drawing session.

# MVSETUP on the Model Tab

## Quick Reference

When the *TILEMODE* system variable is on (the default), the following prompt is displayed:

Enable paper space? [No/Yes] <Y>: *Enter* **n** *or press* ENTER

Pressing ENTER turns off TILEMODE and proceeds as described in the following section, MVSETUP on a Layout Tab (page 901)

Entering **n** displays the following prompt:

Enter units type [Scientific/Decimal/Engineering/Architectural/Metric]: *Enter an option*

A list of available units and prompts for the scale factor and paper size are displayed.

Enter the scale factor: *Enter a value*
Enter the paper width: *Enter a value*
Enter the paper height: *Enter a value*

A bounding box is drawn and the command ends.

# MVSETUP on a Layout Tab

## Quick Reference

When the *TILEMODE* system variable is off, or when you enter **y** or press ENTER at the Enable Paper Space prompt, the following prompt is displayed:

Enter an option [Align (page 901)/Create (page 902)/Scale viewports (page 905)/Options (page 905)/Title block (page 906)/Undo (page 907)]: *Enter an option or press* ENTER *to end the command*

### Align

Pans the view in a viewport so that it aligns with a base point in another viewport. The current viewport is the viewport that the other point moves to.

Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo]: *Enter an option*

**Angled** Pans the view in a viewport in a specified direction.

Specify base point: *Specify a point*

Specify point in viewport to be panned: *Specify a point in the viewport to be panned*

The next two prompts specify the distance and angle from the base point to the second point.

Specify the distance and angle to the new alignment point in the current viewport where you specified the base point.

Specify distance from base point: *Specify a distance*

Specify angle from base point: *Specify an angle*

**Horizontal** Pans the view in one viewport until it aligns horizontally with a base point in another viewport. This option should be used only if the two viewports are oriented horizontally. Otherwise, the view might be panned outside the limits of the viewport.

Specify base point: *Specify a point*

Specify point in viewport to be panned: *Specify a point in the viewport to be panned*

**Vertical Alignment** Pans the view in one viewport until it aligns vertically with a base point in another viewport. This option should be used only if the two viewports are oriented vertically. Otherwise, the view might be panned outside the limits of the viewport.

Specify base point: *Specify a point*

Specify point in viewport to be panned: *Specify a point in the viewport to be panned*

**Rotate View** Rotates the view in a viewport about a base point.

Specify base point in the viewport with the view to be rotated: *Specify a point*

Specify angle from base point: *Specify an angle*

**Undo** Reverses operations performed in the current MVSETUP session.

### Create

Creates viewports.

Enter an option [Delete objects/Create viewports/Undo] <Create>: *Enter an option or press* ENTER

### Delete Objects

Deletes existing viewports.

Select the objects to delete...
Select objects: *Select the viewports to delete and press*ENTER

### Create Viewports

Displays options for creating viewports.

Available layout options:
0: None
1: Single
2: Std. Engineering
3: Array of Viewports
Enter layout number to load or [Redisplay]: *Enter an option number (***0**-**3***), or enter* **r** *to redisplay the list of viewport layout options*
**Layout Number to Load** Controls creation of viewports.

Entering **0** or pressing ENTER creates no viewports.

Entering **1** creates a single viewport whose size is determined by the following prompts.

Specify first corner of bounding area for viewport(s): *Specify a point for the first corner*

Opposite corner: *Specify a point for the opposite corner*

Entering **2** creates four viewports by dividing a specified area into quadrants. You are prompted for the area to be divided and the distance between the viewports.

Specify first corner of bounding area for viewport(s): *Specify a point for the first corner*

Opposite corner: *Specify a point for the opposite corner*

Specify distance between viewports in X direction <0.0>: *Specify a distance or press* ENTER

Specify distance between viewports in Y direction <0.0>: *Specify a distance or press* ENTER

The viewing angle for each quadrant is set as shown in the table.

**Standard engineering viewports**

| Quadrant | View |
|---|---|
| Upper-left | Top (*XY* plane of UCS) |
| Upper-right | SE isometric view |
| Lower-left | Front (*XZ* plane of UCS) |
| Lower-right | Right side (*YZ* plane of UCS) |

Entering **3** defines a matrix of viewports along the *X* and *Y* axes. Specifying points at the next two prompts defines the rectangular area of the drawing that contains the viewport configuration. If you have inserted a title block, the Specify First Corner prompt also includes an option for selecting a default area.

Specify first corner of bounding area for viewport(s): *Specify a point for the first corner*

Opposite corner: *Specify a point for the opposite corner*

Enter number of viewports in X direction <1>: *Enter the number of viewports to place along the* X *axis*

Enter number of viewports in Y direction <1>: *Enter the number of viewports to place along the* Y *axis*

If you enter more than one viewport in each direction, the following prompts are displayed:

Specify distance between viewports in X direction <0.0>: *Specify a distance*

Specify distance between viewports in Y direction <0.0>: *Specify a distance*

The array of viewports is inserted into the defined area.

**Redisplay** Redisplays the list of viewport layout options.

### Undo

Reverses operations performed in the current MVSETUP session.

### Scale Viewports

Adjusts the zoom scale factor of the objects displayed in the viewports. The zoom scale factor is a ratio between the scale of the border in paper space and the scale of the drawing objects displayed in the viewports.

Select the viewports to scale...
Select objects:*Select the viewports to scale*

If you select only one viewport, the next prompt is skipped.

Set zoom scale factors for viewports. Interactively/<Uniform>: *Enter* **i** *or press* ENTER
Enter the number of paper space units <1.0>: *Enter a value or press* ENTER
Enter the number of model space units <1.0>: *Enter a value or press* ENTER
**Interactively** Selects one viewport at a time and displays the following prompts for each:

Enter the number of paper space units <1.0>: *Enter a value or press* ENTER

Enter the number of model space units <1.0>: *Enter a value or press* ENTER

For example, for an engineering drawing at a scale of 1:4, or quarter scale, enter **1** for paper space units and **4** for model space units.

**Uniform** Sets the same scale factor for all viewports.

Number of paper space units. <1.0>: *Enter a value or press* ENTER

Number of model space units. <1.0>: *Enter a value or press*ENTER

### Options

Sets the MVSETUP preferences before you change your drawing.

Enter an option [Layer/LImits/Units/Xref] <exit>: *Enter an option or press* ENTER *to return to the previous prompt*
**Layer** Specifies a layer on which to insert the title block.

Enter layer name for title block or [. (for current layer)]: *Enter an existing or a new layer name, enter a period (.) for the current layer, or press* ENTER

**Limits** Specifies whether to reset the grid limits to the drawing extents after a title block has been inserted.

Set drawing limits? [Yes/No] <N>: *Enter* **y** *or press* ENTER

**Units** Specifies whether the sizes and point locations are translated to inch or millimeter paper units.

Enter paper space units type [Feet/Inches/MEters/Millimeters] <*current*>: *Enter an option or press* ENTER

**Xref** Specifies whether the title block is inserted or externally referenced.

Enter title block placement method [Xref attach/Insert] <*current*>: *Enter* **x**, *enter* **i**, *or press* ENTER

### Title Block

Prepares paper space, orients the drawing by setting the origin, and creates a drawing border and a title block.

Enter title block option [Delete objects/Origin/Undo/Insert] <Insert>: *Enter an option or press* ENTER

### Delete Objects

Deletes objects from paper space.

Select the objects to delete . . .
Select objects: *Use an object selection method*

### Origin

Relocates the origin point for this sheet.

Specify new origin point for this sheet: *Specify a point*

### Undo

Reverses operations performed in the current MVSETUP session.

### Insert

Displays title block options.

Available title blocks:...
0: None
1: ISO A4 Size(mm)
2: ISO A3 Size(mm)
3: ISO A2 Size(mm)
4: ISO A1 Size(mm)
5: ISO A0 Size(mm)
6: ANSI-V Size(in)
7: ANSI-A Size(in)
8: ANSI-B Size(in)
9: ANSI-C Size(in)
10: ANSI-D Size(in)

11: ANSI-E Size(in)
12: Arch/Engineering (24 x 36in)
13: Generic D size Sheet (24 x 36in)
Enter number of title block to load or [Add/Delete/Redisplay]: *Enter an option number (***0** through **13***) or enter an option*

**Title Block to Load** Inserts a border and a title block. Entering **0** or pressing ENTER inserts no border. Entering **1** through **13** creates a standard border of the appropriate size. The list includes ANSI and DIN/ISO standard sheets.

**Add** Adds title block options to the list. Selecting this option prompts you to enter the title block description to be displayed in the list and the name of a drawing to insert.

Enter title block description: *Enter a description*

Enter drawing name to insert (without extension): *Enter a file name*

Define default usable area? [Yes/No] <Y>: *Enter* **n** *or press* ENTER

Pressing ENTER displays the following prompts:

Specify lower-left corner: *Specify a point*

Specify upper-right corner: *Specify a point*

A line similar to the following example is added after the last entry in the *mvsetup.dfs* default file:

```
A/E (24 x 18in),arch-b.dwg,(1.12 0.99 0.00),(18.63 17.02 0.00),in
```

The last field of the line specifies whether the title block has been created in inches or in millimeters. The units field allows title blocks created in either unit system to be changed by setting the unit type using the Options option.

You can also add title blocks that have variable attributes.

**Delete** Removes entries from the list.

Enter number of entry to delete from list: *Enter the number of the entry to delete*

**Redisplay** Redisplays the list of title block options.

### Undo

Reverses operations performed in the current MVSETUP session.

# N Commands

# 14

**In this chapter**

- NETLOAD
- NEW
- NEWSHEETSET

# NETLOAD

## Quick Reference

Loads a .NET application

**netload**

The Choose .NET Assembly dialog box, a standard file selection dialog box (page 931), is displayed.

When *FILEDIA* is set to 0 (zero), NETLOAD displays the following command prompt:

Assembly file name: *Enter a file name and press* ENTER.

# NEW

## Quick Reference

Creates a new drawing

File ➤ NewAt the Command prompt, enter new.
**new**

The behavior of the NEW command is determined by the *STARTUP* system variable.

- *1:* NEW displays the Create New Drawing dialog box (page 911).
- *0:* NEW displays the Select Template dialog box (a standard file selection dialog box (page 931)).

If the FILEDIA system variable is set to 0 instead of 1, a command prompt is displayed (page 917). If you set *FILEDIA* to 0, this prompt is displayed regardless of the Startup setting.

# Create New Drawing Dialog Box

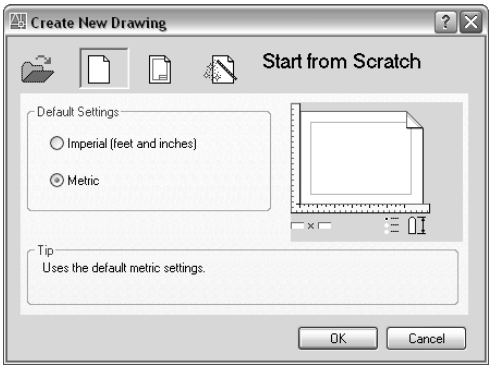
## Quick Reference

File ➤ NewAt the Command prompt, enter new.
**new**

Defines the settings for a new drawing. Start from Scratch creates a new drawing using either imperial or metric default settings. Use a Template creates a new drawing using the settings defined in a drawing template you select. Use a Wizard creates a new drawing using the settings you specify in either the Quick or Advanced wizard. (The first option, Open a Drawing, is not available from the NEW command. To open an existing drawing, use *OPEN*.)

### Start from Scratch

Starts an empty drawing using default imperial or metric settings (*MEASUREINIT* system variable). You can change the measurement system for a given drawing by using the *MEASUREMENT* system variable. The *Drawing1.dwg* that opens when you start the program is a drawing that is started from scratch.




**Imperial** Starts a new drawing based on the imperial measurement system. The default drawing boundary (the grid limits) is 12 by 9 inches.

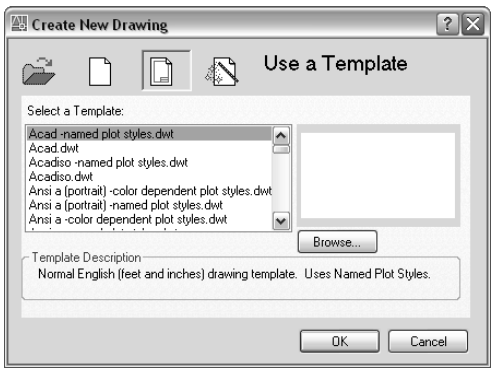
**Metric** Starts a new drawing based on the metric measurement system. The default drawing boundary (the grid limits) is 429 by 297 millimeters.

**Tip** Displays a description of the selected measurement setting.

### Use a Template

Starts a drawing based on a drawing template file. Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the *.dwt* file extension. They are normally kept in the *template* directory.

Several template drawings are included with this program. You can make additional template drawings by changing the extensions of drawing file names to *.dwt*. See "Use a Template File to Start a Drawing" in the *User's Guide*.




**Select a Template** Lists all DWT files that currently exist in the drawing template file location, which is specified in the Options dialog box (page 946). Select a file to use as a starting point for your new drawing.

**Preview** Displays a preview image of the selected file.

**Browse** Displays the Select a Template File dialog box (a standard file selection dialog box (page 931)), where you can access template files that are not available in the Select a Template list.

**Template Description** Displays a description of the selected template. If you create your own template, you can use the Template Options dialog box (page 1242)to specify the text that you want to display here. See the SAVEAS command.

### Use a Wizard

Sets up a drawing using a step-by-step guide. You can choose from two wizards: Quick Setup and Advanced Setup.


**Quick Setup** Displays the Quick Setup wizard (page 913), in which you can specify the units and area for your new drawing. The Quick Setup wizard also changes settings, such as text height and snap spacing, to an appropriate scale.

**Advanced Setup** Displays the Advanced Setup wizard (page 915), in which you can specify the units, angle, angle measure, angle direction, and area for your new drawing. The Advanced Setup wizard also changes settings, such as text height and snap spacing, to an appropriate scale.

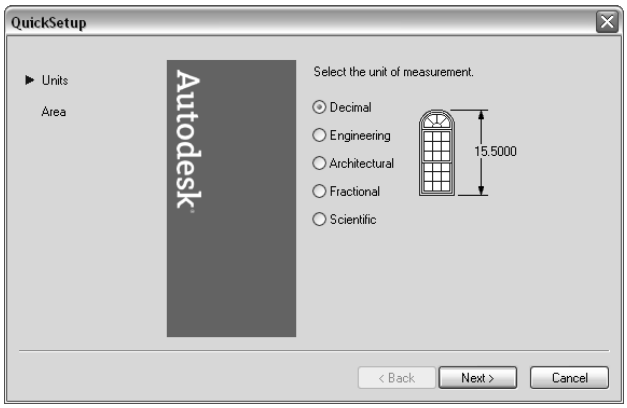
**Wizard Description** Displays a description of the selected wizard.

# Quick Setup Wizard

## Quick Reference

Standard
File ➤ NewAt the Command prompt, enter new.
**new**

Defines the units and area of your drawing.




The Quick Setup wizard has two pages: Units and Area. As you work in this wizard, choose Back and Next to switch between pages; choose Finish on the last page to close the wizard and create the new drawing with the settings you specified.

### Units

Indicates the format in which you enter and in which the program displays coordinates and measurements. Several formats are available. Two of them, Engineering and Architectural, have a specific base unit (inches) assigned to them. You can select from other measurement styles that can represent any convenient unit of measurement.

---

**NOTE** You control the precision (the number of decimal places displayed in all measurements) by using the Advanced Setup wizard or the UNITS command. The default precision used by Quick Setup is four (0.0000).

---

**Decimal** Displays measurements in decimal notation.

**Engineering** Displays measurements in feet and decimal inches.

**Architectural** Displays measurements in feet, inches, and fractional inches.

**Fractional** Displays measurements in mixed-number (integer and fractional) notation.

**Scientific** Displays measurements in scientific notation (numbers expressed in the form of the product of a decimal number between 0 and 10 and a power of 10).

### Area

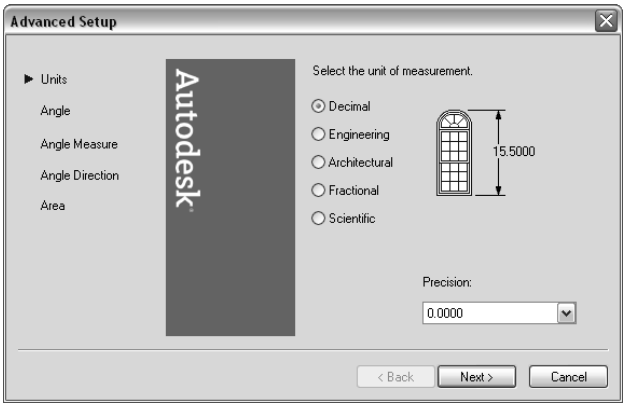
Indicates the width and length in full-scale units of what you plan to draw. This setting limits the area of the drawing covered by grid dots when the grid is turned on. When limits checking is turned on with the *LIMITS* command, this setting also restricts the coordinates you can enter to within the rectangular area. You can change the drawing area and turn limits checking on and off with the LIMITS command.

# Advanced Setup Wizard

## Quick Reference

Standard
File ➤ NewAt the Command prompt, enter new.
**new**

Defines the units, angle, angle measure, angle direction, and area of your new drawing.




The Advanced Setup wizard contains five pages: Units, Angle, Angle Measure, Angle Direction, and Area. As you work in this wizard, choose Back and Next to navigate between pages; choose Finish on the last page to close the wizard and create the new drawing with the settings you specified.

You can later change units, angle, angle measure, and angle direction using the *UNITS* command, and you can change area using the *LIMITS* command.

### Units

Indicates the unit format and precision. The unit format is the format in which you enter and in which the program displays coordinates and measurements. The Units page of the Quick Setup wizard includes the same formats. See Units (page 914) for more information.

The unit precision specifies the number of decimal places or the fractional size for displaying linear measurements.

### Angle

Indicates the format in which you enter angles and in which the program displays angles.

**Decimal Degrees** Displays partial degrees as decimals.

**Deg/Min/Sec** Displays partial degrees as minutes and seconds.

**Grads** Displays angles as grads.

**Radians** Displays angles as radians.

**Surveyor** Displays angles in surveyor's units.

### Angle Measure

Indicates the direction of the 0 angle for the entry of angles. When you enter an angle value, the angle is measured either counter-clockwise or clockwise from the compass direction that you select on this page. You control the counterclockwise/clockwise direction on the Angle Direction page.

**East** Specifies the compass direction east as the 0 angle.

**North** Specifies the compass direction north as the 0 angle.

**West** Specifies the compass direction west as the 0 angle.

**South** Specifies the compass direction south as the 0 angle.

**Other** Specifies a direction other than east, north, west, or south. Enter a specific compass angle to treat as the 0 angle.

### Angle Direction

Indicates the direction from the 0 angle in which you enter and in which the program displays positive angle values: counterclockwise or clockwise.

**Area**

Indicates the width and length in full-scale units of what you plan to draw. This setting limits the area of the drawing covered by grid dots when the grid is turned on. When limits checking is turned on with the *LIMITS* command, this setting also restricts the coordinates you can enter to within the rectangular area. You can change the drawing area and turn limits checking on and off with the LIMITS command.

## NEW Command Line

### Quick Reference

If you turned off the display of the Create New Drawing dialog box, or if *FILEDIA* is set to 0, NEW displays the following prompt:

Enter template file name or [. (for none)] <*current*>: *Enter a name, enter a period (.), or press* ENTER

Enter a tilde (~) at the prompt to display the Select Template dialog box (a standard file selection dialog box (page 931)).

# NEWSHEETSET

### Quick Reference

Creates a new sheet set

File ➤ New Sheet SetDoes not appear in the menus
**newsheetset**

The Create Sheet Set wizard contains a series of pages that step you through the process of creating a new sheet set. You can choose to create a new sheet set from existing drawings, or use an existing sheet set as a template on which to base your new sheet set.

# O Commands

# 15

**In this chapter**

- OBJECTSCALE
- OFFSET
- OLELINKS
- OLESCALE
- OOPS
- OPEN
- OPENDWFMARKUP
- OPENSHEETSET
- OPTIONS
- ORTHO
- OSNAP

# OBJECTSCALE

## Quick Reference

Adds or deletes supported scales for objects

Modify ➤ Annotative Object Scale ➤ Add/Delete ScalesDoes not exist in the menus.
**objectscale** (or '**objectscale** for transparent use)
Annotation Scaling panel; Add Current Scale, Delete Current Scale, and Add/Delete Scales
With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add/Delete Scales.

If you enter **objectscale**, you are prompted to select annotative objects.

Select annotative objects: *Use an object selection method*

The Annotative Object Scale Dialog Box (page 920) is displayed.

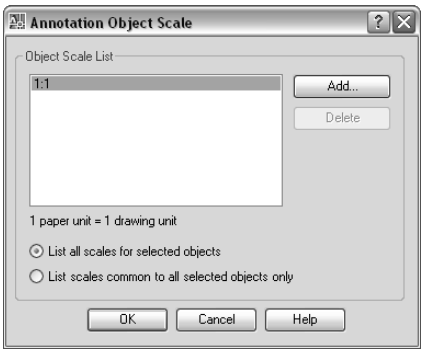
If you enter **-objectscale** at the command prompt, options are displayed at the command prompt (page 922).

# Annotative Object Scale Dialog Box

## Quick Reference

Modify ➤ Annotative Object Scale ➤ Add/Delete ScalesDoes not exist in the menus.
**objectscale** (or '**objectscale** for transparent use)
With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add/Delete Scales.

Adds or deletes supported scales for the selected object.




**Object Scale List** Displays the list of scales supported by the selected object.

**Scale in Units (Unlabeled)** Displays the scale (in units) of the named scale selected in the Object Scale List.

**List all scales for selected objects** Specifies that all scales supported by the selected objects are displayed in the Object Scale List.

**List scales common to all selected objects only** Specifies that only the supported scales that are common to all selected objects are displayed in the Object Scale List.

**Add** Displays the Add Scales to Object Dialog Box (page 921).

**Delete** Removes the selected scale from the scale list.

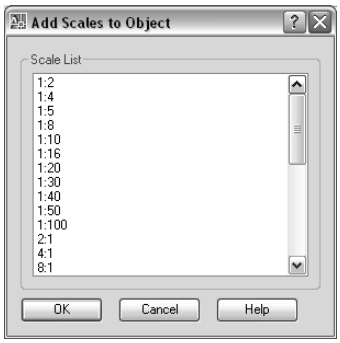
**NOTE** The current scale or scales referenced by objects or views cannot be deleted.

# Add Scales to Object Dialog Box

## Quick Reference

Modify ➤ Annotative Object Scale ➤ Add/Delete ScalesDoes not exist in the menus.
**objectscale** (or '**objectscale** for transparent use)
With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add/Delete Scales.

Adds a new scale to the selected object.




**Scale List** Displays the list of scales that can be added to the selected annotative object. Multiple scales can be selected by holding down the SHIFT or CTRL key while selecting the scales.

Use the SCALELISTEDIT (page 1247) command to add custom scales to this list.

# -OBJECTSCALE

## Quick Reference

If you enter **-objectscale** at the command prompt, the following OBJECTSCALE command prompts are displayed.

Select annotative objects: *Use an object selection method*
Enter an option [Add/Delete] <Add>: *Enter* **a** *or* **d** *or press* ENTER

### Add

Adds a scale to the selected objects.

Enter named scale to add or [?] <*current annotation scale* >: *Enter a descriptive or numeric name such as 1"=4' or 1:48.*
**?** Displays the list of scales in the scale list.

### Delete

Removes a specified scale. If an annotative object supports a single scale, that scale cannot be deleted from the object.

Enter named scale to delete or [?] <*current annotation scale*>: *Enter the name of a scale that you want to delete*
**?** Displays the union of all scales of the selected objects.

# OFFSET

## Quick Reference

Creates concentric circles, parallel lines, and parallel curves

Modify 

Modify ➤ OffsetAt the Command prompt, enter offset.

**offset**

Current settings: Erase source = *current* Layer = *current* OFFSETGAPTYPE = *current*

Specify offset distance (page 923) or [Through (page 924)/Erase (page 924)/Layer (page 924)] <*current*>:*Specify a distance, enter an option, or press* ENTER


The OFFSET command repeats for convenience. To exit the command, press ENTER.

### Offset Distance

Creates an object at a specified distance from an existing object.

Select object to offset or [Exit/Undo]<exit>: *Select one object, enter an option, or press* ENTER *to end the command*

Specify point on side to offset or [Exit/Multiple/Undo] <exit or next object>: *Specify a point (1) on the side of the object you want to offset or enter an option*


**Exit** Exits the OFFSET command.

**Multiple** Enters the Multiple offset mode, which repeats the offset operation using the current offset distance.

**Undo** Reverses the previous offset.

### Through

Creates an object passing through a specified point.

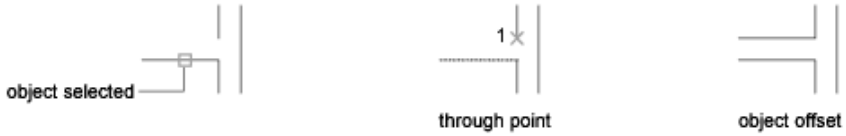
Select object to offset or <exit>: *Select one object or press* ENTER *to end the command*

---

**NOTE** For best results when you offset a polyline with corners, specify the through point near the midpoint of a line segment, not near a corner.

---

Specify through point or [Exit/Multiple/Undo]<exit or next object>: *Specify a point (1) through which you want the offset object to pass or enter a distance*




**Exit** Exits the OFFSET command.

**Multiple** Enters the Multiple offset mode, which repeats the offset operation and accepts additional through points.

**Undo** Reverses the previous offset.

### Erase

Erases the source object after it is offset.

Erase source object after offsetting? [Yes/No]<*current*>: *Enter* **y** *or* **n**

### Layer

Determines whether offset objects are created on the current layer or on the layer of the source object.

Enter layer option for offset objects [Current/Source]<*current*>: *Enter an option*

# OLELINKS

## Quick Reference

Updates, changes, and cancels existing OLE links

Edit ➤ OLE LinksAt the Command prompt, enter olelinks.
**olelinks**

The Links dialog box (page 925) is displayed. If there is no existing OLE link in the drawing, OLE Links is not available on the Edit menu and the Links dialog box is not displayed. To specify a different source application for an embedded object, right-click the embedded object and click Convert on the shortcut menu to display the Convert dialog box (page 926).

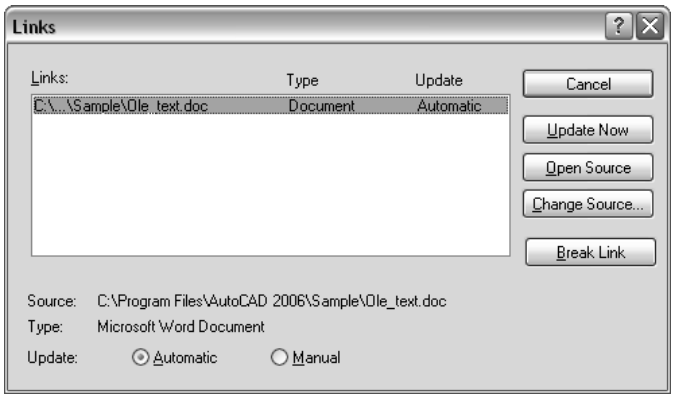
See "Link and Embed Data (OLE)" in the *User's Guide*.

# Links Dialog Box

## Quick Reference

Edit ➤ OLE LinksAt the Command prompt, enter olelinks.
**olelinks**

Lists and modifies linked files and objects.



**Links** Lists information about linked objects. The information listed depends on the type of link. To change information for a linked object, select the object.

**Source** Displays the path name of the source file and the type of object.

**Type** Displays the format type.

**Update: Automatic** Updates the link automatically whenever the source changes.

**Update: Manual** Prompts you to update a link when you open the document.

**Update Now** Updates the selected links.

**Open Source** Opens the source file and highlights the portion linked to the AutoCAD drawing.

**Change Source** Displays the Change Source dialog box (a standard file dialog box), in which you can specify a different source file. If the source is a selection within a file (instead of the entire file), Item Name displays a string representing the selection.

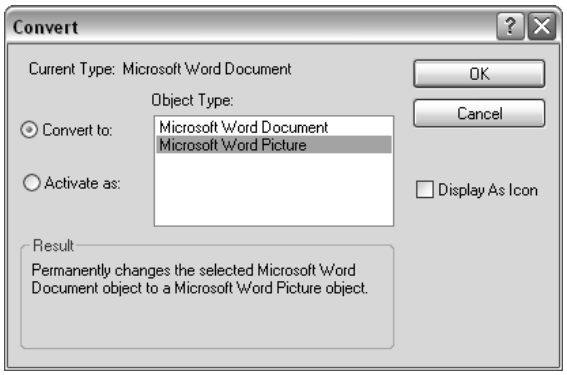
**Break Link** Severs the link between the object and the original file. The object in your drawing is changed to WMF (Windows metafile format), which is not affected by future changes to the original file.

## Convert Dialog Box

### Quick Reference

Right-click an embedded object and click OLE ➤ Convert.

Specifies a different source application for an embedded object.




**Current Type** Displays the type of object you are converting or activating.

**Object Type** Displays a list of available object types. If you select AutoCAD Entities, text is converted to a multiline text (mtext) object, a spreadsheet is converted to a table object, and a bitmap file is converted to an image object.

**Convert To** Converts the embedded object to the type specified under Object Type.

**Activate As** Opens the embedded object as the type selected under Object Type but returns the object to its current type after editing.

**Display as Icon** Displays the source application's icon in the drawing. Double-clicking the icon displays the linked or embedded information.

**Change Icon** Displays the Change Icon dialog box (page 927). This option is available only if you select Display as Icon.

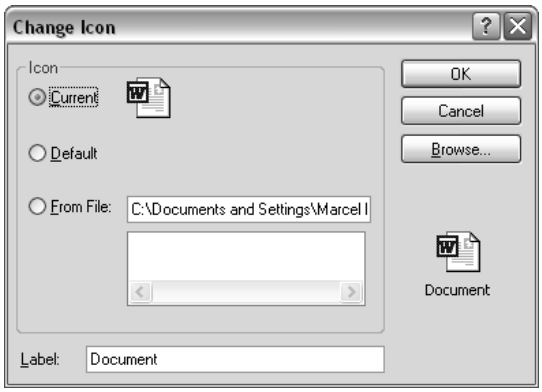
**Result** Describes the result of the selected options.

# Change Icon Dialog Box

## Quick Reference

Edit ➤ OLE LinksAt the Command prompt, enter olelinks.
**olelinks**

Changes the icon that represents an embedded or linked object in a drawing.




#### Icon

Specifies an icon to represent an embedded or linked object.

**Current** Specifies the currently selected icon.

**Default** Specifies the default icon for the source application.

**From File** Specifies an icon from a file (valid types include EXE, DLL, and ICO).

#### Label

Specifies a caption for the icon. The file type is displayed unless you specify otherwise.

#### Browse

Displays the Browse dialog box (a standard file selection dialog box), in which you can select an icon from a file.

# OLESCALE

## Quick Reference

Controls the size, scale, and other properties of a selected OLE object

With an OLE object selected, right-click and click Text Size.
**olescale**

The OLE Text Size dialog box (page 929) is displayed.

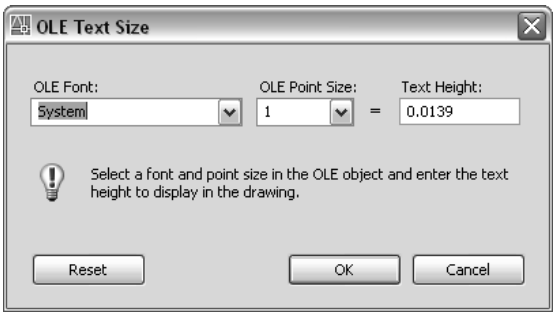
**NOTE** You must select an OLE object before entering the OLESCALE command.

# OLE Text Size Dialog Box

## Quick Reference

With an OLE object selected, right-click and click Text Size.
**olescale**

Maps the point size of one of the fonts in an OLE object to a text height in the drawing. The OLE object is scaled automatically to accommodate the size of the text.




**OLE Font** Displays a list of the fonts used in the OLE object.

**OLE Point Size** Displays a list of the point sizes available for the selected font.

**Text Height** Sets a text height for the font at the selected point size.

**Reset** Restores the OLE object to its size when it was inserted in the drawing.

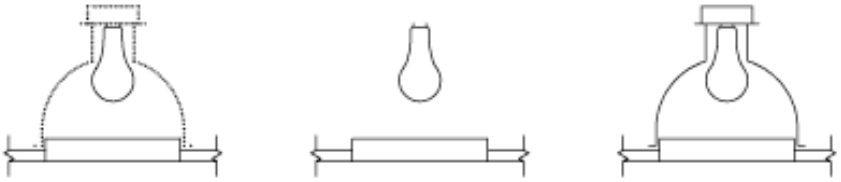
# OOPS

## Quick Reference

Restores erased objects

**oops**

OOPS restores objects erased by the last *ERASE* command.




You can also use OOPS after *BLOCK* or *WBLOCK* because these commands can erase the selected objects after creating a block. However, you cannot use OOPS to restore parameters, actions, or grips in the Block Editor (page 160).

You cannot use OOPS to restore objects on a layer that has been removed with the *PURGE* command.

# OPEN

## Quick Reference

Opens an existing drawing file

Standard 
File ➤ OpenAt the Command prompt, enter open.
**open**

The Select File dialog box (a standard file selection dialog box (page 931)) is displayed.

You can open and load a portion of a drawing, including geometry on a specific view or layer. In the Select File dialog box, click the arrow next to Open and choose Partial Open or Partial Open Read-Only to display the Partial Open dialog box (page 942).

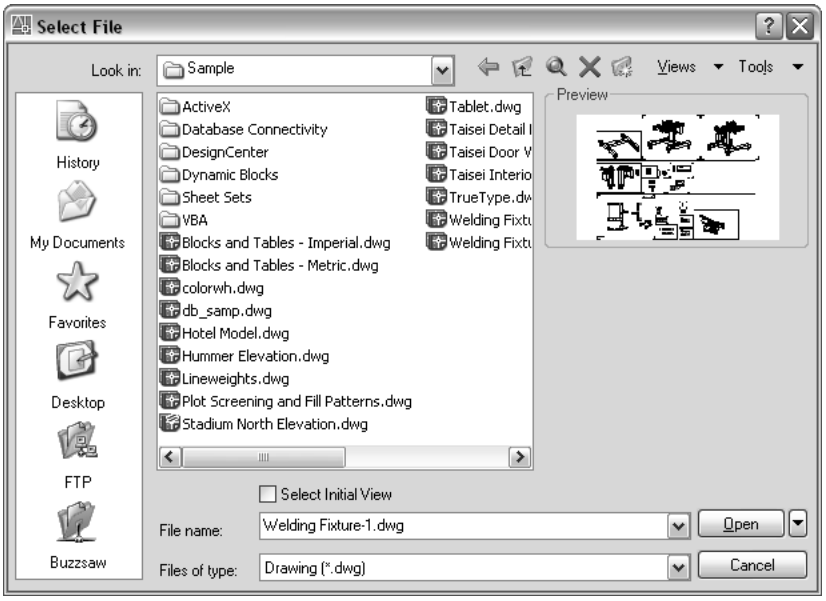
When *FILEDIA* is set to 0 (zero), OPEN displays a command prompt (page 945).

**NOTE** If the drawing you open contains macros, the AutoCAD Macro Virus Protection dialog box (page 1492) is displayed.

# Standard File Selection Dialog Boxes

## Quick Reference

Several commands display standard file selection dialog boxes, in which you can navigate through local and network drives and through FTP sites and Web folders to select files. While each dialog box may vary slightly, following is a list of the possible options.




### Places List

Provides quick access to predefined locations. You can reorder the icons in the Places list by dragging them to a new position. To add a new icon to Places, drag a folder from the list. Right-click in Places to display a shortcut menu with options for adding, removing, and modifying icons or restoring default icons that have been removed. Changes to Places affect all standard file selection dialog boxes.

**History** Displays shortcuts to the files most recently accessed from the dialog box. It is recommended that you periodically remove unwanted shortcuts from the History list. Select History, then select the unwanted shortcuts in

the Files list and click Delete. To sort the shortcuts by date, click Views ➤ Details, and then click the Modified column in the Files list.

**Personal/My Documents** Displays the contents of the *Personal* or *My Documents* folder for the current user profile. The name of this location ("Personal" or "My Documents") depends on your operating system version.

**Favorites** Displays the contents of the *Favorites* folder for the current user profile. This folder consists of shortcuts to files or folders that you added to Favorites using the Tools ➤ Add to Favorites option in the dialog box.

**FTP** Displays the FTP sites that are available for browsing in the standard file selection dialog box. To add FTP locations to this list, or to modify an existing FTP location, click Tools ➤ Add/Modify FTP Locations (page 940) in the dialog box.

**Desktop** Displays the contents of your desktop.

**Buzzsaw** Provides access to Buzzsaw projects. Buzzsaw is a secure Internet-based collaboration and project management service that connects project teams in different locations. To use Buzzsaw, you must already have a project hosting account on Buzzsaw or be given access to a subscriber's Buzzsaw site. If you do not yet have a project hosting account on Buzzsaw, clicking Buzzsaw displays the Select Template dialog box where you can add a Buzzsaw location shortcut. Double-clicking Add a Buzzsaw Location Shortcut opens the Log In To Buzzsaw dialog box where you can register for a free 30-day trial subscription to the service. Also, the first time you access Buzzsaw, your default web browser opens with the project hosting page displayed. If you already have a project hosting account, clicking Buzzsaw shows all of your project sites in the Site list.

### Look In/Save In

Displays the current folder or drive. Click the arrow to view the hierarchy of the folder path and to navigate up the path tree or to other drives, network connections, FTP locations, or web folders (either *Web Folders* or *My Network Places,* depending on the version of your operating system). You can create web folders in Windows Explorer. Consult your system administrator or Windows Explorer Help for more information about web folders. To control whether the last used paths in each particular standard file selection dialog box are stored across sessions, use the REMEMBERFOLDERS system variable.

### Back

Returns to the previous file location.

### Up One Level

Navigates one level up in the current path tree.

### Search the Web

Displays the Browse the Web dialog box, from which you can access and store files on the Internet.

### Delete

Deletes the selected file or folder.

### Create New Folder

Creates a new folder in the current path using a name that you specify.

### Views

Controls the appearance of the Files list or Folders list and, for a Files list, specifies whether to show a preview image when a file is selected.

**List** Displays a multicolumn list.

**Details** Displays a single-column list with file details.

**Preview** Displays a bitmap of the selected file. The Preview area is blank if you do not select a file. To save a bitmap with a drawing file, use the Save a Thumbnail Preview Image option on the Open and Save tab in the Options dialog box.

### Tools

Provides tools to help with file selection and other actions available in file selection dialog boxes.

**Find** Displays the Find dialog box (page 938), in which you can search for files using name, location, and date-modified filters.

**Locate** Uses the AutoCAD search path to locate the file specified in File Name. You set the search path on the Files tab in the Options dialog box (page 946).

**Add/Modify FTP Locations** Displays the Add/Modify FTP Locations dialog box (page 940), in which you can specify the FTP sites to make available for browsing. To browse these sites, select FTP from the Places list.

**Add Current Folder to Places** Adds an icon for the selected folder to the Places list, providing quick access to that folder from all standard file selection dialog boxes. To remove the icon, right-click the icon and choose Remove.

**Add to Favorites** Creates a shortcut to the current Look In location, or to the selected file or folder. The shortcut is placed in the *Favorites* folder for the current user profile, which you can access by choosing Favorites in the Places list.

**Look in Favorites** Sets the Look In path to your system's *Favorites* folder.

**Options** Provides additional options for certain standard file selection dialog boxes.

**Security Options** Provides options for a digital signature and password that are invoked when you save a file.

### Files List

Displays the files and folders in the current path and of the selected file type. Use the Views menu in the dialog box to switch between List view and Details view.

### Preview

Displays a bitmap of the selected file when you choose Views ➤ Preview in the dialog box. The Preview area is blank if you do not select a file. To save a bitmap with a drawing file, use the Save a Thumbnail Preview Image option on the Open and Save tab in the Options dialog box (page 946).

### File Name

Displays the name of the file you select in the Files list. If you select multiple files, File Name displays each selected file within quotation marks. If you enter a file name in the File Name box, any selection is cleared. You must use quotation marks when entering multiple file names. You can use wild-card characters to filter files displayed in the Files list.

### Files of Type

Filters the list of files by file type. When you are saving files, Files of Type specifies the format in which the file is saved.

#### Select Initial View

Displays the specified model space view when you open the drawing if the drawing contains more than one named view.

#### Update Sheet and View Thumbnails Now

Reflects the current setting of the *UPDATETHUMBNAIL* system variable. This option temporarily overrides that setting. Clear the checkbox to save the file without updating thumbnails. Check the box to update all thumbnails. This option does not change the current setting of UPDATETHUMBNAIL.

#### Open/Save

Depending on the purpose of the specific file selection dialog box, opens or saves the selected file or enters the path of the selected folder in the previous dialog box. Certain file selection dialog boxes may include additional options, accessed by clicking the arrow next to the Open button.

**Open Read-Only** Opens a file in read-only mode. You cannot save changes to the file using the original file name.

**Partial Open** Displays the Partial Open dialog box (page 942). You can open and load a portion of a drawing, including geometry on a specific view or layer. *PARTIALOPEN* cannot be used with a drawing that is not in the latest drawing file format.

**Partial Open Read-Only** Opens the specified drawing portions in read-only mode.

## Buzzsaw Location Shortcuts Dialog Box

### Quick Reference

Creates, renames, modifies, and deletes Buzzsaw location shortcuts.

**Buzzsaw Location Shortcuts** Lists the location of existing Buzzsaw shortcuts.

**New** Opens the Log In to Buzzsaw Site dialog box.

**Rename** Renames the shortcuts you select.

**Modify** Opens the Log In to Buzzsaw Site dialog box, where you can change settings for the selected shortcut.

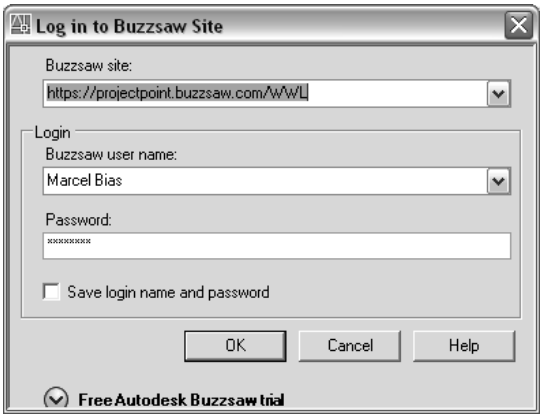
**Delete** Removes the selected shortcut.

**Close** Closes the selected shortcut location.

# Log In to Buzzsaw Dialog Box

## Quick Reference

Logs in to the Buzzsaw site in order to browse to a specific location




**Buzzsaw Site** Specifies the location of the most recently used Buzzsaw site. Enter any existing address or site name.

**Buzzsaw User Name** Specifies the user name you use for the Buzzsaw site you want to access.

**Password** Specifies your password for the Buzzsaw site.

**Save Login Name and Password** Saves your user and login name so that you don't need to retype it the next time you access the site.

**Click here to register for a free trial** Registers you for a free trial of the Buzzsaw site.

# Create a Buzzsaw Location Shortcut Dialog Box

## Quick Reference

Sets a shortcut to a Buzzsaw location.

Provides access to a new Autodesk Buzzsaw location that you specify. You can open drawings or save drawings at this location. If a sheet set is open, the Autodesk Buzzsaw location is associated with your current sheet set. The only Buzzsaw locations displayed are those associated with the current sheet set.

**Buzzsaw Site** Displays the name of the current Buzzsaw site.

**Select a Buzzsaw Site Project or Folder** Lists the most recently opened projects and folders in the site.

**Browse** Displays the Select a Buzzsaw Project or Folder dialog box, which you can use to find and select a project or folder.

**Enter a Name for this Buzzsaw Location Shortcut** Specifies the name of the shortcut.

# Edit a Buzzsaw Location Shortcut Dialog Box

## Quick Reference

Modfies a shortcut to a Buzzsaw location.

Provides edit access to an Autodesk Buzzsaw location that you specify. If a sheet set is open, the Autodesk Buzzsaw location is associated with your current sheet set. The only Buzzsaw locations displayed are those associated with the current sheet set.

**Buzzsaw Site** Displays the name of the current Buzzsaw site.

**Select a Buzzsaw Site Project or Folder** Lists the most recently opened projects and folders in the site.

**Browse** Displays the Select a Buzzsaw Project or Folder dialog box, which you can use to find and select a project or folder.

**Enter a Name for this Buzzsaw Location Shortcut** Specifies the name of the shortcut.

# Select a Buzzsaw Location Dialog Box

## Quick Reference

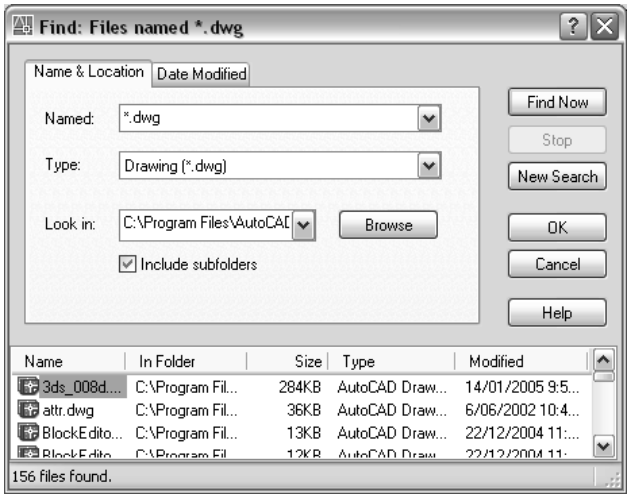
Specifies a Buzzsaw project or folder to save with a Buzzsaw location shortcut.

**List of Buzzsaw Projects and Folders** Specifies Buzzsaw projects or folders that can be linked to the shortcut.

# Find Dialog Box

## Quick Reference

Searches for files using name, location, and date modified filters. To access the Find dialog box, in a standard file selection dialog box (page 931), click Tools menu ➤ Find.




### Name & Location Tab

Filters the file search based on file type, file name, and location.

**Named** Specifies the full or partial file names to include in the search. When entering multiple file names, enclose each name in quotation marks. Use the following wild-card characters to broaden your search:

- ** (Asterisk):* Matches any string and can be used anywhere in the search string.
- *? (Question mark):* Matches any single character; for example, ?BC matches ABC, 3BC, and so on.

The type of files to include in the search is determined by Type, so you don't need to enter a file extension in Named. For example, to find *house102.dwg*, enter **house*** and set Type to Drawing (**.dwg*).

**Type** Specifies the type of files to include in the search. Available file types are limited to those supported by the specific standard file selection dialog box.

**Look In** Specifies the location to search. Enter a location, choose a location from the list, or choose Browse to navigate to a location. When entering multiple locations, enclose each location in quotation marks.

**Browse** Displays the Browse for Folder dialog box, in which you can navigate to a location for the search.

**Include Subfolders** Searches all subfolders within the Look In location.

### Date Modified Tab

Filters the file search based on the file's creation or modified date.

**All Files** Searches without a date filter.

**Find All Files Created or Modified** Restricts the search to files created or modified within a specified time.

- *Between:* Searches for files created or modified between two dates. Enter the dates or click the arrows to choose dates from a calendar. To navigate the calendar, use the left and right arrows, or click the month and year.
- *During the Previous N Months:* Searches for files created or modified within a specified number of months.
- *During the Previous N Days:* Searches for files created or modified within a specified number of days.

#### Find Now

Searches for files using the criteria specified in both tabs.

#### Stop

Ends the search.

#### New Search

Clears the search results and resets the default values for all search options.

#### OK

When one or more files are selected in the search results, closes the Find dialog box and enters the selected file names in the standard file selection dialog box.

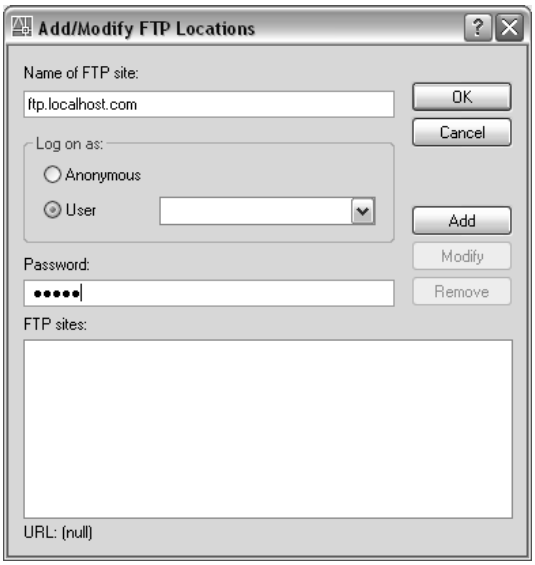
#### Search Results

Lists all files found by the search. To change the sort order of the files, click a column heading. To pass a file to the standard file selection dialog box, select a file and choose OK. When used with a standard file selection dialog box that allows multiple file selection, you can pass multiple files from the Search Results.

## Add/Modify FTP Locations Dialog Box

### Quick Reference

Defines the FTP sites that you can browse within standard file selection dialog boxes. To browse the sites that you add, select FTP from the Places list in the standard file selection dialog box.




### Name of FTP Site

Specifies the site name for the FTP location (for example, *ftp.autodesk.com*).

### Log On As

Specifies whether to log on to the FTP site anonymously or with a specific user name.

**Anonymous** Logs you on to the FTP site as an anonymous user. If the FTP site does not allow anonymous logons, select User and enter a valid user name.

**User** Logs you on to the FTP site using the specified user name.

### Password

Specifies the password to use to log on to the FTP site.

### Add

Adds a new FTP site to the list of FTP locations available from the standard file selection dialog box.

#### Modify

Modifies the selected FTP site to use the specified site name, logon name, and password.

#### Remove

Removes the selected FTP site from the list of FTP locations available from the standard file selection dialog box.

#### FTP Sites

Lists the FTP sites that are available from the Places list in all standard file selection dialog boxes.

#### URL

Displays the URL for the selected FTP site.

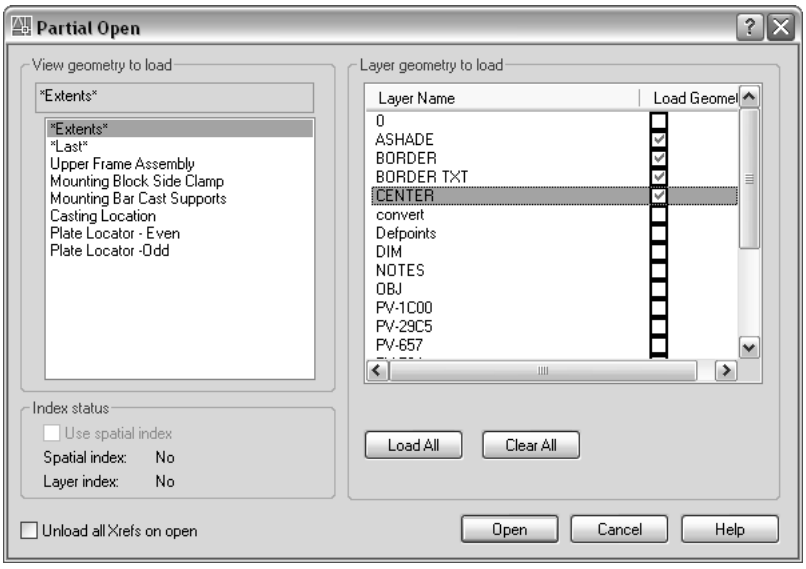
## Partial Open Dialog Box

### Quick Reference

Standard
File ➤ OpenAt the Command prompt, enter open.
**open**

Displays the drawing views and layers available for specifying what geometry to load into the selected drawing. When working with large drawing files, you can select the minimal amount of geometry you need to load when opening a drawing.

When a drawing is partially open, all named objects, as well as the specified geometry, are loaded into the file. Named objects include blocks, dimension styles, layers, layouts, linetypes, text styles, UCSs, views, and viewport configurations.




### View Geometry to Load

Displays the selected view and available views in the drawing. Only model space views are available for loading. You can load paper space geometry by loading the layer that the paper space geometry is drawn on.

**View Name** Displays the currently selected view. Geometry that is common to both the selected view and the layers is loaded into the drawing.

**View List** Displays all the model space views available in the selected drawing file. When a view is selected, the program loads only the geometry in the selected view. The default view is Extents. Geometry that is common to both the selected view and the layers is loaded into the drawing. You can load geometry from only one view.

You can use the *PARTIALOAD* command to load additional geometry into a partially open drawing.

---

**NOTE** When a drawing is partially open, named objects are still loaded into the file along with the specified geometry from the selected view. All views are still available in the partially open drawing, but only geometry from the view specified to load is displayed in the drawing.

---

### Layer Geometry to Load

Displays all the layers available in the selected drawing file. The geometry on selected layers is loaded into the drawing, including both model space and paper space geometry. The default loads no layer geometry into the drawing, but you can load geometry from one or more layers. If you specify to load no layer geometry into the drawing, no geometry from the selected view is loaded either because no layer geometry in that view or in the entire drawing is loaded. Xref-dependent layers are displayed in the Layer Geometry to Load list only if the selected drawing was last saved with the *VISRETAIN* system variable set to 1. Any layers created in the xref since the xref was loaded into the selected drawing are not displayed in the Layer Geometry to Load list.

You can use the *PARTIALOAD* command to load additional geometry into a partially open drawing.

---

**NOTE** When a drawing is partially open, named objects are still loaded into the file along with the specified geometry from the selected layers. All layers are still available in the partially open drawing, but only geometry from the layers specified to load appears in the drawing.

---

**Layer Name** Displays the layer names in the selected drawing.

**Load Geometry** Loads geometry from the layer when the drawing is opened.

**Load All** Loads geometry from all layers when the drawing is opened. You can right-click and use the shortcut menu to load geometry from all layers into the drawing.

**Clear All** Loads no geometry from any layer when the drawing is opened. No geometry at all is loaded, including geometry specified to load from a view. You can right-click and use the shortcut menu to load no geometry into the drawing.

### Index Status

Displays whether the selected drawing file contains a spatial or layer index. *INDEXCTL* controls whether layer and spatial indexes are saved with the drawing file.

**Use Spatial Index** Controls whether a spatial index is used when partially opening a drawing. A spatial index can be used to locate what portion of the drawing is read; this minimizes the time required to open the drawing. If a drawing does not contain a spatial index, this option is unavailable.

**Spatial Index** Displays whether the selected drawing file contains an index that organizes objects based on their location in space.

**Layer Index** Displays whether the selected drawing file contains a list of the objects that are on each layer. A layer index can be used to locate what portion of the drawing is read; this minimizes the time required to open the drawing.

#### Unload All Xrefs on Open

Unloads all external references when opening the drawing. Selecting this option enhances the opening process. If you partially open a drawing that contains a bound xref, only the portion of the xref that is loaded (defined by the selected view) is bound to the partially open drawing.

#### Open

Opens the drawing file, loading only combined geometry from the selected view and layers.

## OPEN Command Line

### Quick Reference

When *FILEDIA* is set to 0 (zero), OPEN displays the following command prompt.

Enter name of drawing to open:

Enter ~ (tilde) at the prompt to ignore FILEDIA and display the Select File dialog box (page 931), a standard file selection dialog box.

# OPENDWFMARKUP

### Quick Reference

Opens a DWF file that contains markups

File ➤ Load Markup SetAt the Command prompt, enter opendwfmarkup.
**opendwfmarkup**

Displays the Open Markup DWF dialog box (a standard file selection dialog box), in which you can select a Design Web Format™ (DWF™) file that contains markups to load into the Markup Set Manager (page 777). When you select a DWF file and click Open, the Markup Set Manager is displayed. The markup set from the selected DWF file is loaded into the Markup Set Manager.

# OPENSHEETSET

### Quick Reference

Opens a selected sheet set

File ➤ Open Sheet SetAt the Command prompt, enter opensheetset.
**opensheetset**

The Open Sheet Set dialog box (a standard file selection dialog box (page 931)) is displayed.

In this dialog box, you can select a sheet set data (DST) file to load sheet set information into the Sheet Set Manager.

# OPTIONS

### Quick Reference

Customizes the program settings

Tools ➤ OptionsAt the Command prompt, enter options.
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

The Options dialog box (page 946) is displayed.

## Options Dialog Box

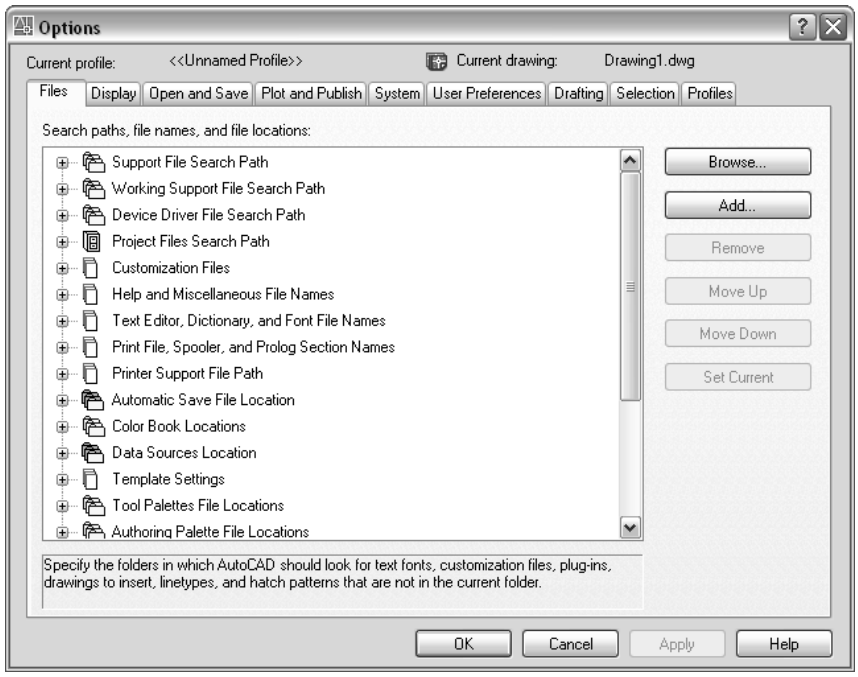
### Quick Reference

Tools ➤ OptionsAt the Command prompt, enter options.

Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.

**options**

Customizes many settings.




**Current Profile** Displays the name of the current profile above the tabs. To set the current profile, create a new profile, or edit an existing profile, use the Profiles tab. (*CPROFILE* system variable)

**Drawing Icon** Signifies that an option is saved with the drawing. An option saved with the drawing affects only the current drawing. An option saved in the registry (and not displayed with a drawing file icon) affects all drawings in a work session. Options that are saved in the registry are saved in the current profile.

**Current Drawing** Displays the name of the current drawing above the tabs. (*DWGNAME* system variable)

The Options dialog box includes the following tabs:

- Files (page 948)
- Display (page 953)

- Open and Save (page 957)
- Plot and Publish (page 961)
- System (page 966)
- User Preferences (page 969)
- Drafting (page 972)
- 3D Modeling (page 975)
- Selection (page 976)
- Profiles (page 980)

# Files Tab (Options Dialog Box)

## Quick Reference

Lists the folders in which the program searches for support, driver, menu, and other files. Also lists optional, user-defined settings such as which dictionary to use for checking spelling.

### Browse

Displays the Browse for Folder or Select a File dialog box (a standard file selection dialog box (page 931)), in which you can specify a new location for the folder or file selected in the list.

### Support File Search Path

Specifies the folders in which the program should look for text fonts, customization files, plug-ins, drawings to insert, linetypes, and hatch patterns that are not in the current folder.

### Working Support File Search Path

Displays the active directories that the program searches for support files specific to your system. The list is read-only and displays valid paths from the Support Files Search Path that exist within the current directory structure and network mappings..

### Device Driver File Search Path

Specifies the search path for device drivers for the video display, pointing devices, printers, and plotters.

### Project Files Search Path

Specifies a project name for the drawing. The project name corresponds to a search path for external reference (xref) files associated with the project. You can create any number of project names with associated folders, but each drawing can have only one project name. (*PROJECTNAME* system variable)

### Customization Files

Specifies the names and locations of various types of files.

**Main Customization File** Specifies the default location of the main customization file (*acad.cui*).

**Enterprise Customization File** Specifies the location of an enterprise customization file.

**Custom Icon Location** Specifies the location for custom icons used in customization files.

### Help and Miscellaneous File Names

Specifies the names and locations of various types of files.

**Help File** Specifies the location of the Help file.

**Default Internet Location** Specifies the default Internet location used by both the *BROWSER* command and the Browse the Web button on the Web toolbar.

**Configuration File** Specifies the location of the configuration file used to store hardware device driver information. This value is read-only and can be changed only by using the /c command line switch. See "Customize Startup" in the *User's Guide*.

### Text Editor, Dictionary, and Font File Names

Specifies a number of optional settings.

**Text Editor Application** Specifies the text editor application to use for editing multiline text objects. (*MTEXTED* system variable)

**Main Dictionary** Specifies the dictionary to use for checking spelling. (*DCTMAIN* system variable)

**Custom Dictionary File** Specifies a custom dictionary to use (if you have one). (*DCTCUST* system variable)

**Alternate Font File** Specifies the location of the font file to use if the original font cannot be located and an alternate font is not specified in the font mapping file. If you click Browse, the Alternate Font dialog box (page 982) is displayed, from which you can choose an available font. (*FONTALT* system variable)

**Font Mapping File** Specifies the location of the file that defines how to convert fonts that cannot be found. (*FONTMAP* system variable)

### Print File, Spooler, and Prolog Section Names

Specifies settings related to plotting.

**Plot File Name for Legacy Plotting Scripts** Specifies a default name for the temporary plot files used with plotting scripts created with AutoCAD Release 14 or earlier. The default name is the *drawing name* plus the *.plt* file extension. The default name used with AutoCAD 2000 and later drawings is the *drawing name-layout name* plus the *.plt* file name extension. Some plotting device drivers, however, use a different plot file extension. This option affects only the default plot file name used for plotting scripts created with earlier versions of AutoCAD.

**Print Spool Executable** Specifies the application to use for print spooling. You can enter the executable file name as well as any command line arguments you want to use. For example, you can enter **myspool.bat %s** to spool plot files to *myspool.bat* and have a unique plot file name automatically generated. See "Use AutoSpool" in the *Driver and Peripheral Guide*.

### Printer Support File Path

Specifies search path settings for printer support files.

**Print Spooler File Location** Specifies the path for print spool files.

**Printer Configuration Search Path** Specifies the path for printer configuration files (PC3 files).

**Printer Description File Search Path** Specifies the path for files with a *.pmp* file extension, or printer description files.

**Plot Style Table Search Path** Specifies the path for files with an *.stb* or *.ctb* extension, or plot style table files (both named plot style tables and color-dependent plot style tables).

### Automatic Save File Location

Specifies the path for the file created when you select Automatic Save on the Open and Save tab. (*SAVEFILEPATH* system variable)

### Color Book Locations

Specifies the path for color book files that can be used when specifying colors in the Select Color dialog box. You can define multiple folders for each path specified. This option is saved with the user profile.

### Data Sources Location

Specifies the path for database source files. Changes to this setting do not take effect until you close and restart the program.

### Template Settings

Specifies the drawing template settings.

**Drawing Template File Location** Specifies the path to locate drawing template files used by the Start Up wizard and New dialog box.

**Sheet Set Template File Location** Specifies the path to locate sheet set template files used by the Create Sheet Set wizard.

**Default Template File Name for QNEW** Specifies the drawing template file used by the *QNEW* command.

**Default Template for Sheet Creation and Page Setup Overrides** Specifies the default template file that is used for creating new sheets and to store page setup overrides that can be applied to Publish operations from the Sheet Set Manager.

### Tool Palettes File Locations

Specifies the path for tool palette support files.

### Authoring Palette File Locations

Specifies the path for the Block Authoring Palettes support files. Block Authoring Palettes are used in the Block Editor and provide tools for creating dynamic blocks.

### Log File Location

Specifies the path for the log file created when you select Maintain a Log File on the Open and Save tab. (*LOGFILEPATH* system variable)

### Plot and Publish Log File Location

Specifies the path for the log file that is created if you select the Automatically Save Plot and Publish Log option on the Plot and Publish tab (page 961).

### Temporary Drawing File Location

Specifies the location to store temporary files. This program creates temporary files and then deletes them when you exit the program. If you plan to run the program from a write-protected folder (for example, if you are working on a network or opening files from a CD), specify an alternate location for your temporary files. The folder you specify must not be write-protected.

The *TEMPPREFIX* system variable (read-only) also stores the current location of temporary drawing files.

### Temporary External Reference File Location

Specifies the location of external reference (xref) files. This location is used for the copy of the xref when you select Enabled with Copy in the Demand Load Xrefs list on the Open and Save tab. (*XLOADPATH* system variable)

### Texture Maps Search Path

Specifies the folders to search for rendering texture maps.

### Web File Search Path

Specifies the folders to search for photometric web files.

#### i-drop Associated File Location

Specifies the location of data files associated with i-drop content. When the location is not specified, the location of the current drawing file is used.

#### Browse

Displays the Browse for Folder or Select a File dialog box, depending on what you selected in the List of Folders and Files.

#### Add

Adds a search path for the selected folder.

#### Remove

Removes the selected search path or file.

#### Move Up

Moves the selected search path above the preceding search path.

#### Move Down

Moves the selected search path below the following search path.

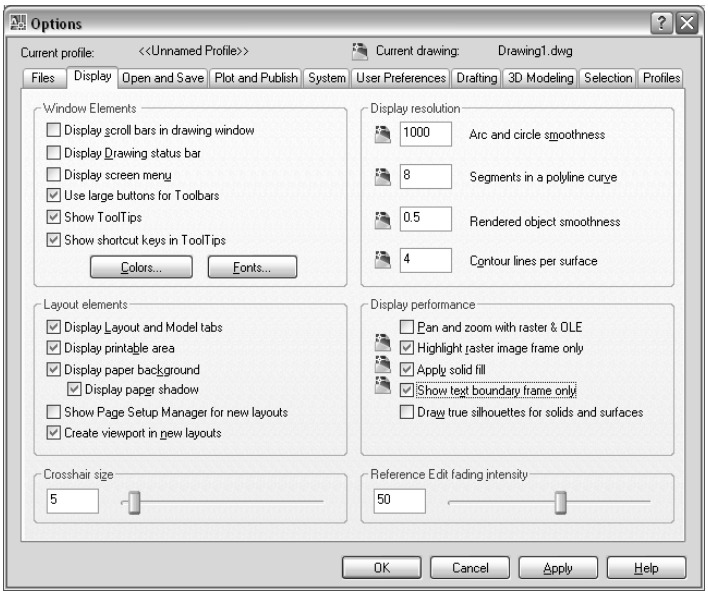
#### Set Current

Makes the selected project or spelling dictionary current.

## Display Tab (Options Dialog Box)

### Quick Reference

Customizes the display.




### Window Elements

Controls display settings specific to the drawing environment.

**Display Scroll Bars in Drawing Window** Displays scroll bars at the bottom and right sides of the drawing area.

**Display Drawing Status Bar** Displays the drawing status bar, which displays several tools for scaling annotations.

When the drawing status bar is turned on it displays at the bottom of the drawing area. When the drawing status bar is turned off, the tools found on the drawing status bar are moved to the application status bar.

**Display Screen Menu** Displays the screen menu on the right side of the drawing area. The screen menu font is controlled by the Windows system font settings. If you use the screen menu, you should set the Windows system font setting to a font and font size that fits the screen menu size restrictions.

**Use Large Buttons for Toolbars** Displays buttons in a larger format at 32 by 30 pixels. The default display size is 16 by 15 pixels.

**Show Tooltips** Displays tooltips when you move the cursor over buttons on the toolbar.

**Show Shortcut Keys in Tooltips** Displays shortcut keys when you move the cursor over buttons on the toolbar.

**Colors** Displays the Color Options dialog box (page 983). Use this dialog box to specify colors of elements in the main application window.

**Fonts** Displays the Command Line Window Font dialog box (page 985). Use this dialog box to specify the font for the command window text.

### Layout Elements

Controls options for existing and new layouts. A layout is a paper space environment in which you can set up drawings for plotting.

**Display Layout and Model Tabs** Displays the layout and Model tabs at the bottom of the drawing area. When this option is cleared, the tabs are replaced by buttons on the status bar.

**Display Printable Area** Displays printable area in a layout. The printable area is represented by the area within the dashed line and is determined by the selected output device. Objects drawn outside of the printable area are clipped or omitted when the drawing is plotted.

**Display Paper Background** Displays a representation of the specified paper size in a layout. The paper size and plot scale determine the size of the paper background.

*Display Paper Shadow:* Displays a shadow around the paper background in a layout. This option is unavailable if the Display Paper Background option is cleared.

**Show Page Setup Manager for New Layouts** Displays the Page Setup Manager (page 1001) the first time you click a layout tab. Use this dialog box to set options related to paper and plot settings.

**Create Viewport in New Layouts** Creates a single viewport automatically when you create a new layout.

### Crosshair Size

Controls the size of the crosshairs. The valid range is from 1 to 100 percent of the total screen. At 100 percent, the ends of the crosshairs are never visible. When the size is decreased to 99 percent or below, the crosshairs have a finite size, and the ends of the crosshairs are visible when situated at the edge of the drawing area. The default size is 5 percent. (*CURSORSIZE* system variable)

### Display Resolution

Controls the quality of the display of objects. If you set high values to improve display quality, the impact on performance is significant.

**Arc and Circle Smoothness** Controls the smoothness of circles, arcs, and ellipses. A higher number produces smoother objects, but more time is required to regenerate, pan, and zoom the objects. You can improve performance by setting this option to a low value such as 100 for drawing, and increasing the value for rendering. The valid range is 1 to 20,000. The default setting is 1000. This setting is saved in the drawing. To change the default for new drawings, consider specifying this setting in the template files on which you base your new drawings. (*VIEWRES* command)

**Segments in a Polyline Curve** Sets the number of line segments to be generated for each polyline curve. The higher the number, the greater the performance impact. Set this option to a low value such as 4 to optimize performance for drawing. Values range from -32767 to 32767. The default setting is 8. This setting is saved in the drawing. (*SPLINESEGS* system variable)

**Rendered Object Smoothness** Controls the smoothness of shaded and rendered curved solids. The value you enter for Rendered Object Smoothness is multiplied by the value you enter for Arc and Circle Smoothness to determine how to display solid objects. To improve performance, set Rendered Object Smoothness to 1 or less when drawing. A higher number decreases display performance and increases rendering time. The valid range is 0.01 to 10. The default setting is 0.5. This setting is saved in the drawing. (*FACETRES* system variable)

**Contour Lines per Surface** Sets the number of contour lines per surface on objects. A higher number decreases display performance and increases rendering time. The valid range is 0 to 2047. The default setting is 4. This setting is saved in the drawing. (*ISOLINES* system variable)

### Display Performance

Controls display settings that affect performance.

**Pan and Zoom with Raster and OLE** Controls the display of raster images and OLE objects when you use Realtime *PAN* and *ZOOM*. Clear this option to optimize performance. If dragging display is turned on and you select Pan and Zoom with Raster and OLE, a copy of the object moves with the cursor as you reposition the original. Dragging display controls whether an outline of the object is displayed while you drag it. The *DRAGMODE* system variable controls dragging display. (*RTDISPLAY* system variable)

**Highlight Raster Image Frame Only** Controls the display of raster images during selection. If this option is selected, only the frame of the raster image is highlighted when selected. Select this option to optimize performance. (*IMAGEHLT* system variable)

**Apply Solid Fill** Displays solid fills in objects. You must regenerate the drawing for this setting to take effect. This setting is saved in the drawing. Clear this option to optimize performance. (*FILLMODE* system variable)

Objects affected by FILL include hatches (including solid-fill), two-dimensional solids, wide polylines, multilines, and traces.

**Show Text Boundary Frame Only** Displays the frames for text objects instead of displaying the text objects. After you select or clear this option, you must use REGEN to update the display. This setting is saved in the drawing. Select this option to optimize performance. (*QTEXTMODE* system variable)

**Draw True Silhouettes for Solids and Surfaces** Controls whether silhouette edges of 3D solid objects are displayed when the current visual style is set to 2D Wireframe or 3D Wireframe. This option also controls whether mesh is drawn or suppressed when a 3D solid object is hidden. This setting is saved in the drawing. Clear this option to optimize performance. (*DISPSILH* system variable)

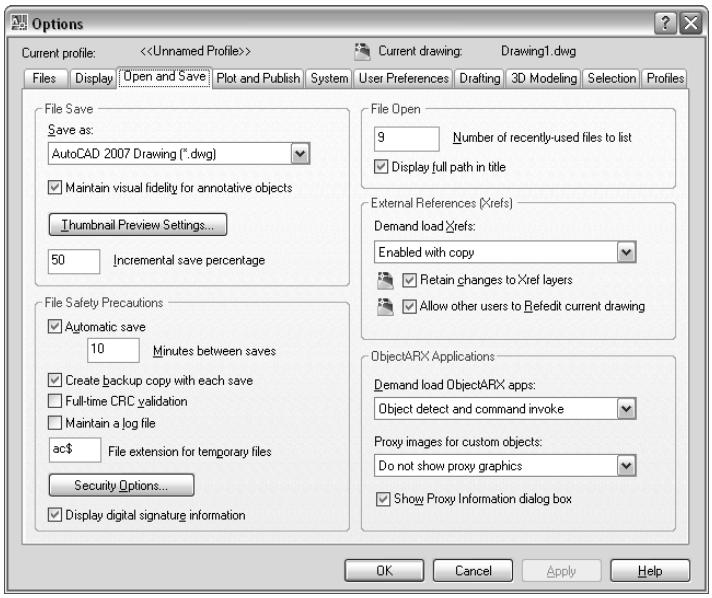
#### Reference Edit Fading Intensity

Specifies the fading intensity value for objects during in-place reference editing. With in-place reference editing, you can edit a block reference or external reference from within the current drawing. While references are being edited in place, objects that are not being edited are displayed at a lesser intensity than objects that can be edited. The valid range is 0 through 90 percent. The default setting is 50 percent. (*XFADECTL* system variable)

## Open and Save Tab (Options Dialog Box)

### Quick Reference

Controls options that relate to opening and saving files.




### File Save

Controls settings related to saving a file.

**Save As** Displays the valid file formats used when saving a file with *SAVE*, *SAVEAS*, *QSAVE*, and *WBLOCK*. The file format selected for this option is the default format that all drawings are saved as when you use SAVE, SAVEAS, QSAVE, and WBLOCK.

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**NOTE** AutoCAD 2004 is the drawing file format used by the AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006 releases.

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**Maintain Visual Fidelity for Annotative Objects** Specifies whether or not drawings are saved with visual fidelity for objects. Click the information icon to learn more about saving drawings with visual fidelity.

Annotative objects may have multiple . Annotative objects are decomposed and scale representations are saved to separate layers, which are named based on their original layer and appended with a number.

**Thumbnail Preview Settings** Displays the Thumbnail Preview Settings dialog box (page 987), which controls whether thumbnail previews are updated when the drawing is saved.

**Incremental Save Percentage** Sets the percentage of potentially wasted space in a drawing file. Full saves eliminate wasted space. Incremental saves are faster but they increase the size of your drawing. If you set Incremental Save Percentage to 0, every save is a full save. For optimum performance, set the value to 50. If hard disk space becomes an issue, set the value to 25. If you set the value to 20 or less, performance of the SAVE and SAVEAS commands slows significantly. (*ISAVEPERCENT* system variable)

### File Safety Precautions

Assists in avoiding data loss and in detecting errors.

**Automatic Save** Saves your drawing automatically at the intervals you specify. You can specify the location of all Autosave files by using the *SAVEFILEPATH* system variable. *SAVEFILE* (read-only) stores the name of the Autosave file.

**NOTE** Automatic save is disabled when the Block Editor (page 160) is open.

*Minutes Between Saves:* When Automatic Save is on, specifies how often the drawing is saved. (*SAVETIME* system variable)

**Create Backup Copy with Each Save** Specifies whether a backup copy of a drawing is created when you save the drawing. The backup copy is created in the same location as the drawing. (*ISAVEBAK* system variable)

For information about using backup files, see "Create and Restore Backup Files" in the *User's Guide*.

**Full-Time CRC Validation** Specifies whether a cyclic redundancy check (CRC) should be performed each time an object is read into the drawing. CRC is an error-checking mechanism. If your drawings are being corrupted and you suspect a hardware problem or a software error, turn this option on.

**Maintain a Log File** Writes the contents of the text window to a log file. To specify the location and name of the log file, use the Files tab in the Options dialog box. You can also set the log file location by using the *LOGFILEMODE* system variable. The *LOGFILENAME* system variable (read-only) stores the log file name of the current drawing.

**File Extension for Temporary Files** Specifies a unique extension for temporary save files. The default extension is *.ac$*.

**Security Options** Provides options for a digital signature and password that are invoked when you save a file.

**Display Digital Signature Information** Presents digital signature information when a file with a valid digital signature is opened. (SIGWARN system variable)

### File Open

Controls settings that relate to recently used files and open files.

**Number of Recently Used Files to List** Controls the number of recently used files that are listed in the File menu for quick access. Valid values are 0 to 9.

**Display Full Path In Title** Displays the full path of the active drawing in the drawing's title bar, or in the application window title bar if the drawing is maximized.

### External References (Xrefs)

Controls the settings that relate to editing and loading external references.

**Demand Load Xrefs** Controls demand loading of xrefs. Demand loading improves performance by loading only the parts of the referenced drawing needed to regenerate the current drawing. (*XLOADCTL* system variable)

- *Disabled:* Turns off demand loading.
- *Enabled:* Turns on demand loading and improves performance. Select the Enabled setting to enhance the loading process when working with clipped xrefs that contain a spatial or layer index. When this option is selected, other users cannot edit the file while it is being referenced.
- *Enabled with Copy:* Turns on demand loading but uses a copy of the referenced drawing. Other users can edit the original drawing.

**Retain Changes to Xref Layers** Saves changes to layer properties and states for xref-dependent layers. When the drawing is reloaded, the properties currently assigned to xref-dependent layers are retained. This setting is saved in the drawing. (*VISRETAIN* system variable)

**Allow Other Users to Refedit Current Drawing** Determines whether the current drawing file can be edited in place if it is being referenced by another drawing or multiple drawings. This setting is saved in the drawing. (*XEDIT* system variable)

### ObjectARX Applications

Controls settings that relate to AutoCAD Runtime Extension applications and proxy graphics.

**Demand Load ObjectARX Apps** Specifies if and when a third-party application is demand-loaded if a drawing contains custom objects created in that application. (*DEMANDLOAD* system variable)

- *Disable Load on Demand:* Turns off demand-loading.
- *Custom Object Detect:* Demand-loads the source application when you open a drawing that contains custom objects. This setting does not demand-load the application when you invoke one of the application's commands.
- *Command Invoke:* Demand-loads the source application when you invoke one of the application's commands. This setting does not demand-load the application when you open a drawing that contains custom objects.
- *Object Detect and Command Invoke:* Demand-loads the source application when you open a drawing that contains custom objects or when you invoke one of the application's commands.

**Proxy Images for Custom Objects** Controls the display of custom objects in drawings. (*PROXYSHOW* system variable)

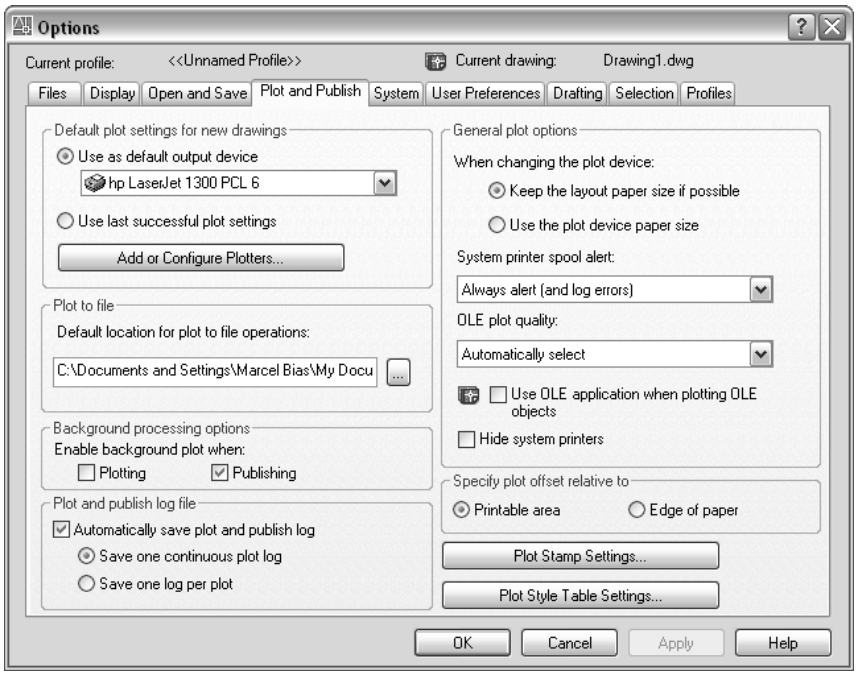
- *Do Not Show Proxy Graphics:* Specifies not to display custom objects in drawings.
- *Show Proxy Graphics:* Specifies to display custom objects in drawings.
- *Show Proxy Bounding Box:* Specifies to show a box in place of custom objects in drawings.

**Show Proxy Information Dialog Box** Specifies whether a warning is displayed when you open a drawing that contains custom objects. (*PROXYNOTICE* system variable)

## Plot and Publish Tab (Options Dialog Box)

### Quick Reference

Controls options related to plotting and publishing.




### Default Plot Settings For New Drawings

Controls default plotting settings for new drawings or drawings created in AutoCAD Release 14 or earlier that have never been saved in AutoCAD 2000 or later format.

**Use As Default Output Device** Sets the default output device for new drawings and for drawings created in AutoCAD Release 14 or earlier that have never been saved in AutoCAD 2000 or later format. The list displays any plotter configuration files (PC3) found in the plotter configuration search path and any system printers that are configured in the system.

**Use Last Successful Plot Settings** Sets the plotting settings to match those of the last successful plot.

**Add or Configure Plotters** Displays the Plotter Manager (page 1079) (a Windows system window). You can add or configure a plotter with the Plotter Manager.

### Plot To File

Specifies the default location for plot to file operations. You can enter a location or click the [...] button to specify a new location.A

### Auto Publish

Specifies whether drawings are published automatically and controls Auto Publish settings.

**Automatic DWF Publish** Specifies whether drawings are published automatically. You can also select this option by setting the *AUTODWFPUBLISH* system variable to 1.

**Automatic DWF Publish Settings** Opens the Publish Options Dialog Box (page 1128)

### Background Processing Options

Specifies options for background plotting and publishing. You can use background plotting to start a job you are plotting or publishing and immediately return to work on your drawing while your job is plotted or published as you work. (*BACKGROUNDPLOT* system variable)

Backround plot and publish details are available through the Plot and Publish status bar icon shortcut menu (page 1518).

You can plot or publish jobs in the background even if the program is not running by entering **acad/pl <DSD file name>** at the Windows Command Prompt.

**Enable Background Plotting When** Specifies whether or not background plotting is turned on for plotting and publishing.

*Plotting:* Specifies that plot jobs are processed in the background. This setting is also affected by the *BACKGROUNDPLOT* system variable.

*Publishing:* Specifies that published jobs are processed in the background. This setting is also affected by the *BACKGROUNDPLOT* system variable.

---

**NOTE** When -PLOT, *PLOT*, -PUBLISH, and *PUBLISH* are used in a script (SCR file), the BACKGROUNDPLOT system variable value is ignored, and -PLOT, PLOT, -PUBLISH, and PUBLISH are processed in the foreground.

---

### Plot and Publish Log File

Controls options for saving a plot and publish log file as a comma-separated value (CSV) file that can be viewed in a spreadsheet program. To view or change the location of the log file, see the Files tab in the Options dialog box (page 948).

This log file contains information about plot and publish jobs such as:

- Job ID
- Job name
- Sheet set name
- Category name
- Date and time started and completed
- Sheet name
- Full file path
- Selected layout name
- Page setup name
- Named page setup path
- Device name
- Paper size name
- Final status

**Automatically Save Plot and Publish Log** Specifies that a log file that contains information about all plot and publish jobs is automatically saved.

**Save One Continuous Plot Log** Specifies that a single log file that contains information about all plot and publish jobs is automatically saved.

**Save One Log File Per Plot** Specifies that a separate log file is created for each plot and publish job.

### General Plot Options

Controls options that relate to the general plotting environment, including paper size settings, system printer alert behavior, and OLE objects in a drawing.

**Keep the Layout Paper Size If Possible** Uses the paper size specified in the Page Setup dialog box (page 1005) as long as the selected output device can plot to this paper size. If the selected output device cannot plot to this paper size, the program displays a warning message and uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also select this option by setting *PAPERUPDATE* to 0.

**Use the Plot Device Paper Size** Uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also set this option by setting PAPERUPDATE to 1.

**System Printer Spool Alert** Controls whether you are alerted when the plotted drawing is spooled through a system printer because of an input or output port conflict.

- *Always Alert (And Log Errors):* Alerts you and always logs an error when the plotted drawing spools through a system printer.
- *Alert First Time Only (And Log Errors):* Alerts you once and always logs an error when the plotted drawing spools through a system printer.
- *Never Alert (And Log First Error):* Never alerts you and logs only the first error when the plotted drawing spools through a system printer.
- *Never Alert (Do Not Log Errors):* Never alerts you or logs an error when the plotted drawing spools through a system printer.

**OLE Plot Quality** Determines the quality of plotted OLE objects. The options are Monochrome, Low Graphics, High Graphics, and Automatically Select. The automatic option assigns a plot quality setting based on the type of OLE object. (*OLEQUALITY* system variable)

**Use OLE Application When Plotting OLE Objects** Launches the application used to create the OLE object when plotting a drawing with OLE objects. You can use this option to optimize the quality of plotted OLE objects. This setting is saved in the drawing. (*OLESTARTUP* system variable)

**Hide System Printer** Controls whether Windows system printers are displayed in the Plot and Page Setup dialog boxes. This option hides standard Windows system printers only. It does not hide Windows system printers that have been configured using the Add-a-Plotter wizard.

You can control the size of the list of devices in the Plot and Page Setup dialog boxes by moving a device's PC3 file out of the Plotters folder and its subfolders.

### Specify Plot Offset Relative To

Specifies whether the offset of the plot area is from the lower-left corner of the printable area or from the edge of the paper (*PLOTOFFSET* system variable).

**Printable Area** Specifies that the plot offset is relative to the printable area.

**Edge of Paper** Specifies that the plot offset is relative to the edge of the paper.

### Plot Stamp Settings

Opens the Plot Stamp dialog box (page 1070).

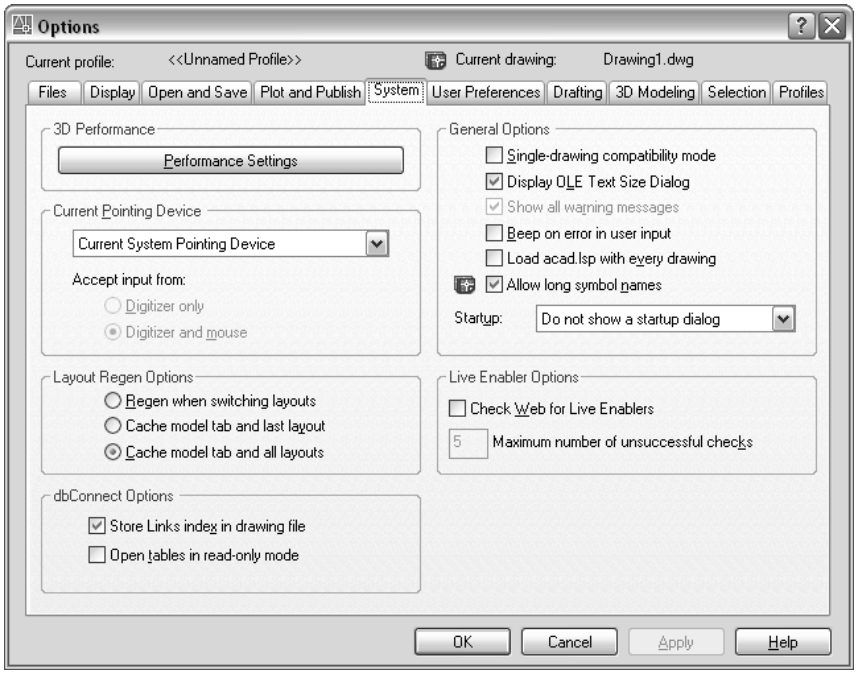
### Plot Style Table Settings

Opens the Plot Style Table Settings dialog box (page 987).

## System Tab (Options Dialog Box)

### Quick Reference

Controls system settings.




### 3D Performance

Controls settings that relate to configuration of the 3D graphics display system.

**3D Performance Settings** Displays the Adaptive Degradation and Performance Tuning dialog box (page 22).

### Current Pointing Device

Controls options that relate to the pointing device.

**Current Pointing Device** Displays a list of the available pointing device drivers.

- *Current System Pointing Device:* Sets the system pointing device as current.
- *Wintab Compatible Digitizer:* Sets the Wintab-compatible digitizer as current.

**Accept Input From** Specifies whether the program accepts input from both a mouse and a digitizer or ignores mouse input when a digitizer is set.

### Layout Regen Options

Specifies how the display list is updated in the Model and layout tabs. For each tab, the display list is updated either by regenerating the drawing when you switch to that tab or by saving the display list to memory and regenerating only the modified objects when you switch to that tab. Changing these settings can improve performance. (LAYOUTREGENCTL system variable)

---

**NOTE** The effectiveness of changing these settings depends on several factors. See *LAYOUTREGENCTL* for more information.

---

**Regen When Switching Layouts** Regenerates the drawing each time you switch tabs.

**Cache Model Tab and Last Layout** For the Model tab and the last layout made current, saves the display list to memory and suppresses regenerations when you switch between the two tabs. For all other layouts, regenerations still occur when you switch to those tabs.

**Cache Model Tab and All Layouts** Regenerates the drawing the first time you switch to each tab. For the remainder of the drawing session, the display list is saved to memory and regenerations are suppressed when you switch to those tabs.

### dbConnect Options

Controls options that relate to database connectivity.

**Store Links Index in Drawing File** Stores the database index in the drawing file. Select this option to enhance performance during Link Select operations. Clear this option to decrease drawing file size and to enhance the opening process for drawings with database information.

**Open Tables in Read-Only Mode** Specifies whether to open database tables in Read-only mode in the drawing file.

### General Options

Controls general options that relate to system settings.

**Display OLE Text Size Dialog** Displays the OLE Text Size dialog box when OLE objects are inserted into drawings.

**Show All Warning Messages** Displays all dialog boxes that include a Don't Display This Warning Again option. All dialog boxes with warning options are displayed, regardless of previous settings specific to each dialog box.

**Beep on Error in User Input** Sounds an alarm beep when an invalid entry is detected.

**Load acad.lsp with Every Drawing** Specifies whether the *acad.lsp* file is loaded into every drawing. If this option is cleared, only the *acaddoc.lsp* file is loaded into all drawing files. Clear this option if you do not want to run certain LISP routines in specific drawing files. (*ACADLSPASDOC* system variable)

**Allow Long Symbol Names** Permits long names to be used for named objects in the drawing definition table. Object names can be up to 255 characters and include letters, numbers, blank spaces, and any special character not used by Windows and this program for other purposes. Long names can be used for layers, dimension styles, blocks, linetypes, text styles, layouts, UCS names, views, and viewport configurations. This option is saved in the drawing. (*EXTNAMES* system variable)

### Live Enabler Options

Specifies whether the program checks for object enablers. Using object enablers, you can display and use custom objects in drawings even when the ObjectARX application that created them is unavailable.

**Check Web for Live Enablers** Checks the Autodesk website for object enablers. (*PROXYWEBSEARCH* system variable)

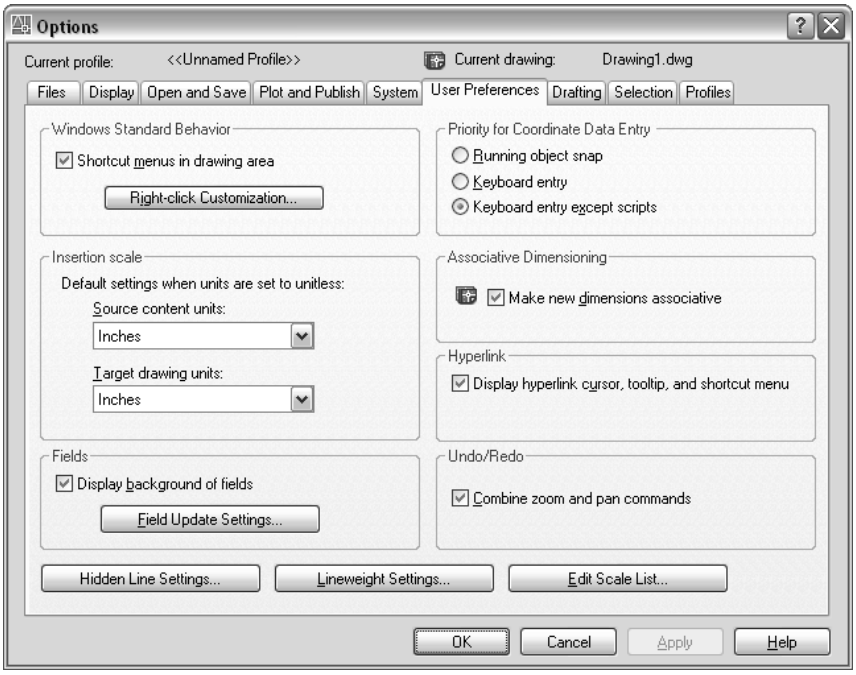
**Maximum Number of Unsuccessful Checks** Specifies the number of times to check for object enablers if unsuccessful.

For more information about object enablers, see “Work with Custom and Proxy Objects” in the *User's Guide*.

# User Preferences Tab (Options Dialog Box)

## Quick Reference

Controls options that optimize the way you work.




### Windows Standard Behavior

Controls keystroke and right-click behavior.

**Double click editing** Controls the double click editing behavior in the drawing area. (*DBLCLKEDIT* system variable)

**Shortcut Menus in Drawing Area** Displays a shortcut menu in the drawing area when you right-click the pointing device. If this option is cleared, right-click is interpreted as ENTER. (*SHORTCUTMENU* system variable)

**Right-Click Customization** Displays the Right-Click Customization dialog box (page 989). This dialog box provides further definition for the Shortcut Menus in Drawing Area option. (*SHORTCUTMENU* system variable)

### Insertion Scale

Controls the default scale for inserting blocks and drawings into a drawing.

**Source Content Units** Sets the units used for an object being inserted into the current drawing when no insert units are specified with the *INSUNITS* system variable. If Unspecified-Unitless is selected, the object is not scaled when inserted. (*INSUNITSDEFSOURCE* system variable)

**Target Drawing Units** Sets the units used in the current drawing when no insert units are specified with the *INSUNITS* system variable. (*INSUNITSDEFTARGET* system variable)

### Fields

Sets preferences related to fields.

**Display Background of Fields** Displays fields with a light gray background that is not plotted. When this option is cleared, fields are displayed with the same background as any text. (*FIELDDISPLAY* system variable)

**Field Update Settings** Displays the Field Update Settings dialog box (page 991). (*FIELDEVAL* system variable)

### Priority for Coordinate Data Entry

Controls how the program responds to input of coordinate data. (*OSNAPCOORD* system variable)

**Running Object Snap** Specifies that running object snaps override coordinate entry at all times. You can also select this option by setting OSNAPCOORD to 0.

**Keyboard Entry** Specifies that coordinate entry overrides running object snaps at all times. You can also select this option by setting OSNAPCOORD to 1.

**Keyboard Entry Except Scripts** Specifies that coordinate entry overrides running object snaps, except in scripts. You can also select this option by setting OSNAPCOORD to 2.

### Associative Dimensioning

Controls whether associative dimension objects or legacy-style, nonassociative dimension objects are created.

**Make New Dimensions Associative** When selected, creates associative dimensions, which automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. (DIMASSOC (page 1655) system variable)

### Hyperlink

Controls settings that relate to the display properties of hyperlinks.

**Display Hyperlink Cursor, Tooltip, and Shortcut Menu** Displays the hyperlink cursor and tooltip whenever the pointing device pauses over an object that contains a hyperlink. The shortcut menu provides additional hyperlink options when you select an object that contains a hyperlink and then right-click in the drawing area. If this option is cleared, hyperlinks in the drawing are ignored. (HYPERLINKOPTIONS (page 652) command)

### Undo/Redo

Controls Undo and Redo for Zoom and Pan.

**Group Zoom and Pan Commands** When selected, groups multiple, consecutive zoom and pan commands as a single action for undo and redo operations.

---

**NOTE** Pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

---

### Lineweight Settings

Displays the Lineweight Settings dialog box (page 770). Use this dialog box to set lineweight options, such as display properties and defaults, and also to set the current lineweight.

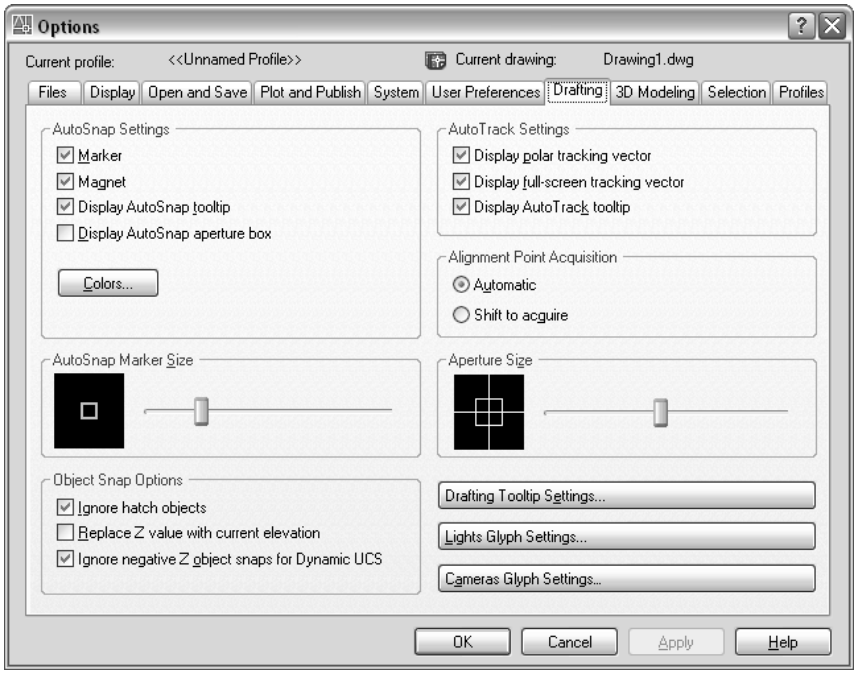
### Edit Scale List

Displays the Edit Scale List dialog box (page 1250). Use this dialog box to manage the list of scales displayed in several dialog boxes associated with layout viewports and plotting.

# Drafting Tab (Options Dialog Box)

## Quick Reference

Sets options for several editing features, including AutoSnap and AutoTrack.




### AutoSnap Settings

Controls settings that relate to the visual aids, called AutoSnaps, that are displayed when you use object snaps. AutoSnap™ settings are saved in the registry. When the cursor, or aperture box, is on an object, you can press TAB to cycle through all the snap points available for that object.

**Marker** Controls the display of the AutoSnap marker. The marker is a geometric symbol that is displayed when the crosshairs move over a snap point. (*AUTOSNAP* system variable)

**Magnet** Turns the AutoSnap magnet on or off. The magnet is an automatic movement of the crosshairs that locks the crosshairs onto the nearest snap point. (AUTOSNAP system variable)

**Display AutoSnap Tooltip** Controls the display of the AutoSnap tooltip. The tooltip is a label that describes which part of the object you are snapping to. (AUTOSNAP system variable)

**Display AutoSnap Aperture Box** Controls the display of the AutoSnap aperture box. The aperture box is a box that appears inside the crosshairs when you snap to an object. (*APBOX* system variable)

**Colors** Displays the Drawing Window Colors dialog box (page 983).

### AutoSnap Marker Size

Sets the display size for the AutoSnap marker.

### Object Snap Options

Specifies options for object snaps. (*OSNAP* command)

**Ignore Hatch Objects** Specifies that object snaps ignore hatch patterns when object snapping is turned on. (*OSOPTIONS* system variable)

**Replace Z Value with Current Elevation** Specifies that object snaps ignore the *Z*-value of the object snap location and use the *Z*-value of the elevation set for the current UCS.

**Ignore Negative Z Object Snaps for Dynamic UCS** Specifies that object snaps ignore geometry with negative *Z* values during use of a dynamic UCS. (*OSOPTIONS* system variable)

### AutoTrack Settings

Controls the settings that relate to AutoTrack™ behavior, which is available when polar tracking or object snap tracking is turned on (see *DSETTINGS*).

**Display Polar Tracking Vector** Displays a vector along specified angles when polar tracking is on. With polar tracking, you can draw lines along angles. Polar angles are 90-degree divisors, such as 45, 30, and 15 degrees.

In a 3D view, a polar tracking vector parallel to the *Z* axis of the UCS is also displayed, and the tooltip displays +Z or -Z for the angle depending on the direction along the Z axis

You can disable Display Polar Tracking Vector by setting *TRACKPATH* to 2.

**Display Full-Screen Tracking Vector** Controls the display of tracking vectors. Tracking vectors are construction lines from which you can draw objects at specific angles or in specific relationships to other objects. If this option is selected, alignment vectors are displayed as infinite lines.

You can disable Display Full-Screen Tracking Vector by setting TRACKPATH to 1.

**Display AutoTrack Tooltip** Controls the display of the AutoTrack and Ortho tooltips. Tooltips are labels that display the tracking coordinates. (*AUTOSNAP* system variable)

### Alignment Point Acquisition

Controls the method of displaying alignment vectors in a drawing.

**Automatic** Displays tracking vectors automatically when the aperture moves over an object snap.

**Shift to Acquire** Displays tracking vectors when you press SHIFT and move the aperture over an object snap.

### Aperture Size

Sets the display size for the AutoSnap aperture. When Display AutoSnap Aperture Box is selected (or when *APBOX* is set to 1), the aperture box is displayed in the center of the crosshairs when you snap to an object. The aperture size determines how close to a snap point you can be before the magnet locks the aperture box to the snap point. Values range from 1 to 50 pixels. (*APERTURE* system variable)

### Drafting Tooltip Appearance

Controls the color, size, and transparency of drafting tooltips.

**Settings** Displays the Tooltip Appearance dialog box (page 500).

### Lights Glyph Settings

Displays the Light Glyph Appearance dialog box (page 985).

### Cameras Glyph Settings

Displays the Camera Glyph Appearance dialog box (page 986).

# 3D Modeling Tab (Options Dialog Box)

## Quick Reference

Sets options for working with solids and surfaces in 3D.

### 3D Crosshairs

Controls settings for the display style of the crosshairs pointer in 3D operations.

**Show Z Axis in Crosshairs** Controls whether the *Z* axis is displayed by the crosshairs pointer.

**Label Axes in Standard Crosshairs** Controls whether axis labels are displayed with the crosshairs pointer.

**Show Labels for Dynamic UCS** Displays axis labels on the crosshairs pointer for the dynamic UCS even when the axis labels are turned off in the Label Axes in Standard Crosshairs box.

**Crosshair Labels** Choose the labels to display with the crosshairs pointer.

**Use X, Y, Z** Labels the axes X, Y, and Z.

**Use N, E, z** Labels the axes with abbreviations for North, East, and Z elevation.

**Use Custom Labels** Labels the axes with characters that you specify.

### Display UCS Icon

Controls the display of the UCS icon.

**Display in 2D Model Space** Displays the UCS icon in model space when the current visual style is set to 2D Wireframe.

**Display in 3D Parallel Projection** Displays the UCS icon in model space when the current visual style is set to 3D Hidden, 3D Wireframe, Conceptual, or Realistic, and the projection style is set to Parallel.

**Display in 3D Perspective Projection** Displays the UCS icon in model space when the current visual style is set to 3D Hidden, 3D Wireframe, Conceptual, or Realistic, and the projection style is set to Perspective.

### Dynamic Input

Controls the display of dynamic input fields for coordinate entry.

**Show Z Field for Pointer Input** Displays a field for the *Z* coordinate when using dynamic input.

### 3D Objects

Controls settings for the display of 3D solids and surfaces.

**Visual Style While Creating 3D Objects** Specifies the visual style while creating 3D solid primitives. (*DRAGVS* system variable)

**Deletion Control While Creating 3D Objects** Specifies whether defining geometry used in creating solids and surfaces is automatically deleted once the 3D object is created or whether you are prompted to delete the objects. (*DELOBJ* system variable)

**U Isolines on Surfaces and Meshes** Sets the isolines property in the U direction for surfaces and meshes. (*SURFU* system variable)

**V Isolines on Surfaces and Meshes** Sets the isolines property in the V direction for surfaces and meshes. (*SURFV* system variable)

### 3D Navigation

Sets walk, fly, and animation options for displaying 3D models.

**Reverse Mouse Wheel Zoom** Reverses the zoom direction on a wheel mouse. (*ZOOMWHEEL* system variable)

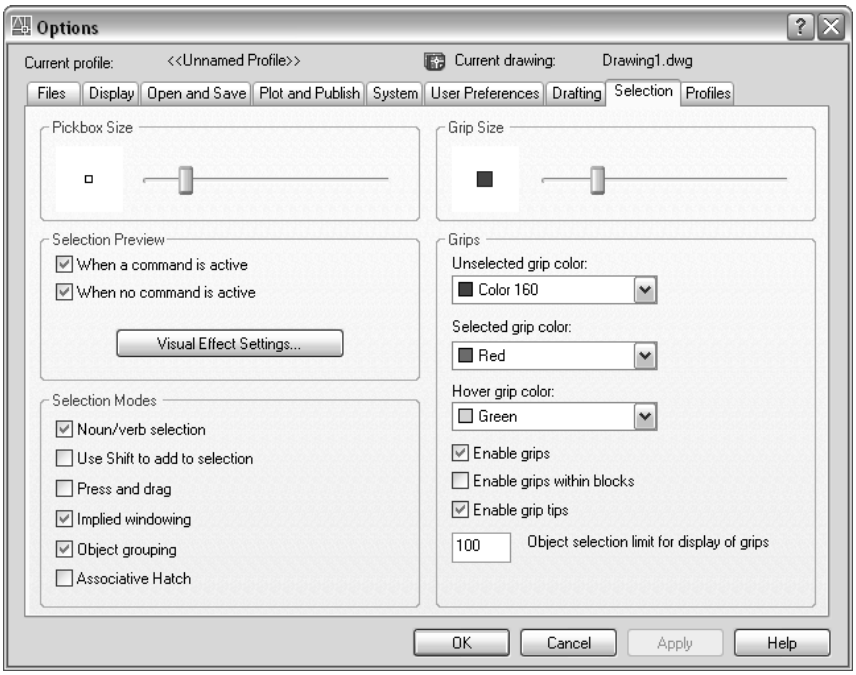
**Walk and Fly Settings** Displays the Walk and Fly Settings dialog box (page 1546).

**Animation Settings** Displays the Animation Settings dialog box (page 54).

## Selection Tab (Options Dialog Box)

### Quick Reference

Sets options for selecting objects.




### Pickbox Size

Controls the display size of the pickbox. The pickbox is the object selection tool that appears in editing commands. (*PICKBOX* system variable)

### Selection Preview

Highlights objects as the pickbox cursor rolls over them. The *PREVIEWEFFECT* system variable controls the appearance of the highlighted objects.

**When a Command Is Active** Displays selection previewing only when a command is active and the Select Objects prompt is displayed. (*SELECTIONPREVIEW* system variable)

**When No Command Is Active** Displays selection previewing when no commands are active. (*SELECTIONPREVIEW* system variable)

**Visual Effect Settings** Displays the Visual Effect Settings dialog box (page 992).

### Selection Modes

Controls settings that relate to object selection methods.

**Noun/Verb Selection** Allows you to select an object before starting a command. The command affects the previously selected object or objects. (*PICKFIRST* system variable)

You can use many editing and inquiry commands with noun/verb selection, including

| | | |
|---|---|---|
| *3DALIGN* | *DVIEW* | *PROPERTIES* |
| *ARRAY* | *ERASE* | *ROTATE* |
| *BLOCK* | *EXPLODE* | *SCALE* |
| *CHANGE* | *LIST* | *STRETCH* |
| *CHPROP* | *MIRROR* | *WBLOCK* |
| *COPY* | *MOVE* | |

**Use SHIFT to Add to Selection** Adds an object to or removes it from the selection set when you press SHIFT and select an object. To clear a selection set quickly, draw a selection window in a blank area of the drawing. (*PICKADD* system variable)

**Press and Drag** Draws a selection window when you select a point and dragging the pointing device to a second point. If this option is cleared, you can draw a selection window by selecting two separate points with the pointing device. (*PICKDRAG* system variable)

**Implied Windowing** Initiates the drawing of a selection window when you select a point outside an object.

Drawing the selection window from left to right selects objects that are entirely inside the window's boundaries. Drawing from right to left selects objects within and crossing the window's boundaries. (*PICKAUTO* system variable)

**Object Grouping** Selects all objects in a group when you select one object in that group. With *GROUP* you can create and name a set of objects for selection. You can also set this option by setting the *PICKSTYLE* system variable to 1.

**Associative Hatch** Determines which objects are selected when you select an associative hatch. If this option is selected, boundary objects are also selected when you select an associative hatch. You can also set this option by setting the *PICKSTYLE* system variable to 2.

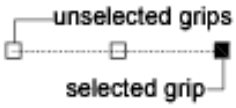
### Grip Size

Controls the display size of grips. (*GRIPSIZE* system variable)


### Grips

Controls the settings that relate to grips. Grips are small squares displayed on an object after it has been selected.

**Unselected Grip Color** Determines the color of an unselected grip. If you choose Select Color from the color list, the Select Color dialog box (page 251) is displayed. An unselected grip is displayed as a small, filled square. (*GRIPCOLOR* system variable)

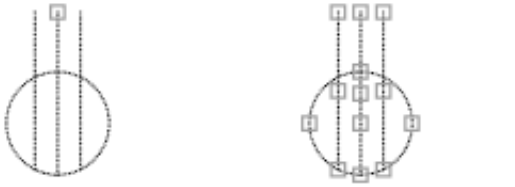



**Selected Grip Color** Determines the color of a selected grip. If you choose Select Color from the color list, the Select Color dialog box (page 251) is displayed. A selected grip is displayed as a small, filled square. (*GRIPHOT* system variable)

**Hover Grip Color** Determines the color a grip displays when the cursor rolls over the grip. If you choose Select Color from the color list, the Select Color dialog box (page 251) is displayed. (*GRIPHOVER* system variable)

**Enable Grips** Displays grips on an object when you select it. You can edit an object with grips by selecting a grip and using the shortcut menu. Displaying grips in a drawing significantly affects performance. Clear this option to optimize performance. (*GRIPS* system variable)

**Enable Grips within Blocks** Controls how grips are displayed on a block after you select it. If this option is selected, all grips are displayed for each object in the block. If this option is cleared, one grip is displayed at the insertion point of the block. You can edit an object with grips by selecting a grip and using the shortcut menu. (*GRIPBLOCK* system variable)



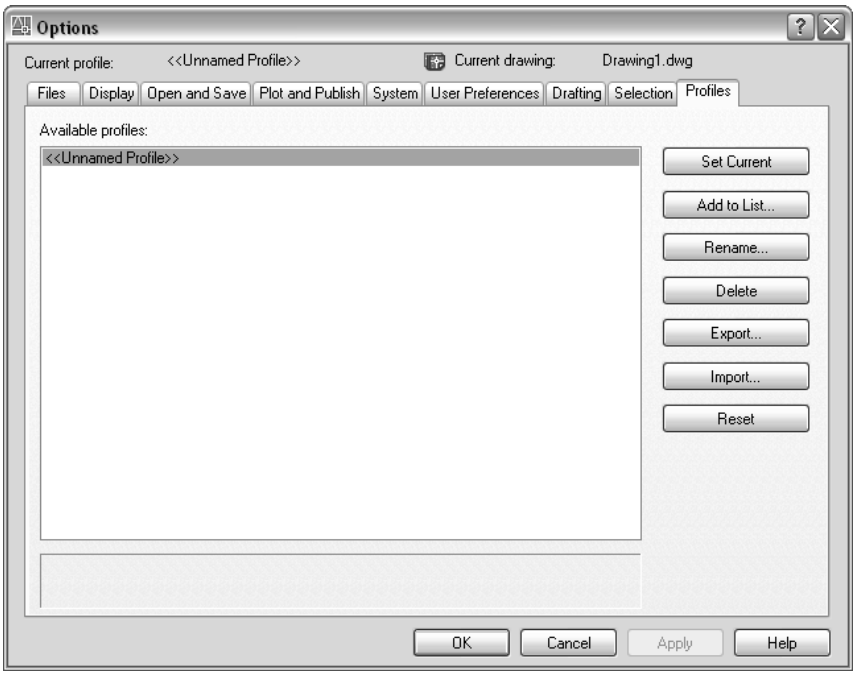

**Enable Grip Tips** Displays grip-specific tips when the cursor hovers over a grip on a custom object that supports grip tips. This option has no effect on standard objects. (*GRIPTIPS* system variable)

**Object Selection Limit for Display of Grips** Suppresses the display of grips when the initial selection set includes more than the specified number of objects. The valid range is 1 to 32,767. The default setting is 100. (*GRIPOBJLIMIT* system variable)

## Profiles Tab (Options Dialog Box)

### Quick Reference

Controls the use of profiles. A profile is a configuration you define.




**Available Profiles** Displays a list of the available profiles. To set the current profile, select a profile and choose Set Current.

**Set Current** Makes the selected profile the current profile.

**Add to List** Displays the Add Profile dialog box (page 994) to save the selected profile under a different name.

**Rename** Displays the Change Profile dialog box (page 994) for changing the name and description of the selected profile. Use Rename when you want to rename a profile but keep its current settings.

**Delete** Deletes the selected profile (unless it is the current profile).

**Export** Exports a profile as a file with an *.arg* extension so the file can be shared with other users. You can import the file on the same computer or a different computer.

**Import** Imports a profile (a file with an *.arg* extension) created by using the Export option.

**Reset** Resets the values in the selected profile to the system default settings.

# Alternate Font Dialog Box

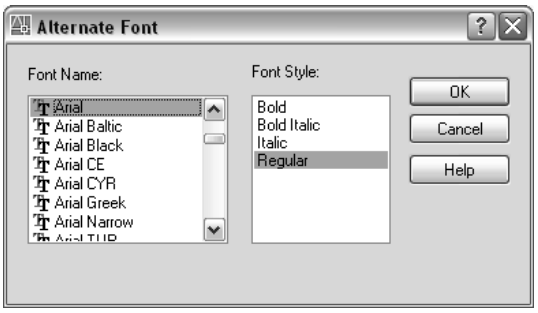
## Quick Reference

Tools ➤ Options (Options dialog box, Files tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.

**options**

Specifies the location of the font file to use if the original font cannot be located and an alternate font is not specified in the font mapping file. You can also set the location of Alternate Font File by using the *FONTALT* system variable.

If you do not specify an alternate font through the Options dialog box or the FONTALT system variable, when you open a drawing containing fonts that cannot be located, a dialog box is displayed in which you specify a font to use for each missing font.




**Font Name** Lists the font family name for all registered TrueType fonts and all SHX fonts in the *Fonts* folder. When you select a name from the list, the program reads the file for the specified font.

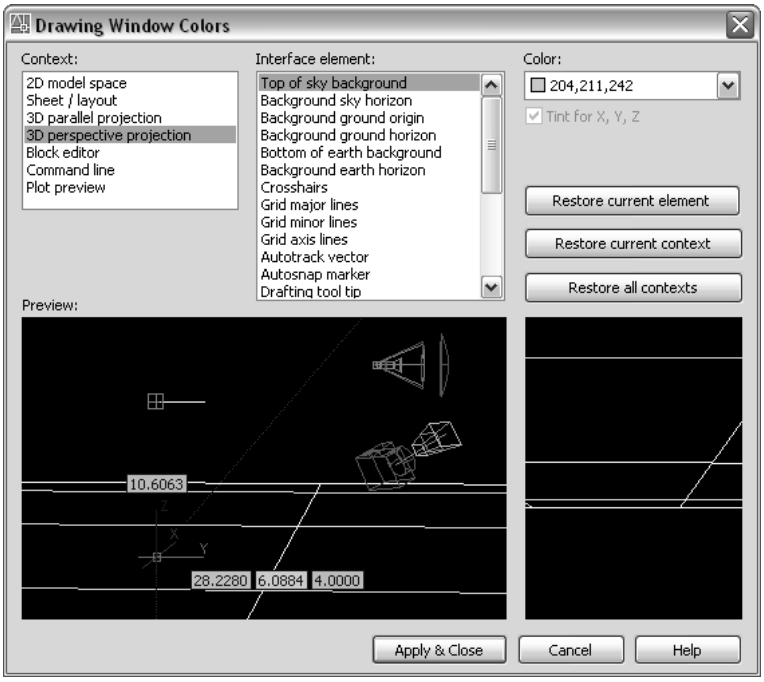
**Font Style** Specifies font character formatting, such as italic, bold, or regular. If the missing font is a Big Font, this option changes to Big Font and lists all available Asian-language Big Font files.

# Drawing Window Colors Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, Display tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and then click Options.
**options**

Sets the display colors of the *interface elements* for each *context* in the application. A context is an operating environment such as model space. An *interface element* is an item that is visible in that context such as the crosshairs pointer or the background color.




### Context

Displays a list of all contexts in the program. A context is an operating environment such as model space. Interface elements can have different colors assigned depending on the context. For example, the background color can be different for model space and paper space. Click the context in the Context

list, click an interface element in the Interface Element list, and then select a color in the Color list. The window title bar and menu bar colors are controlled by the Windows system settings.

### Interface Element

Displays a list of all interface elements in the selected context. An interface element is an item that is visible in a context (for example, the background color). Click the context in the Context list, click an interface element in the Interface Element list, and then select a color in the Color list. The window title bar and menu bar colors are controlled by the Windows system settings.

### Color

Lists the available color settings to apply to the selected interface element. You can choose one of the colors in the color list or choose Select Color to open the Select Color dialog box (page 251).

You can use the Select Color dialog box to define the color of interface elements by selecting from the AutoCAD Color Index (ACI) colors, true color colors, and color book colors.

If you select a new color for an interface element, the new setting is displayed in the Preview area.

**Tint for X, Y, Z** Controls whether *X*, *Y*, and *Z* axis color tinting is applied to the following interface elements: crosshairs pointer, Autotrack vectors, ground plane grid lines, and drafting tooltips. Color tinting applies pure red, blue and green hues using the luminance (lightness) of the color you specify while increasing the color saturation by 50%.

**Restore Current Element** Restores the currently selected interface element to its default color.

**Restore Current Context** Restores all interface elements in the currently selected context to their default colors.

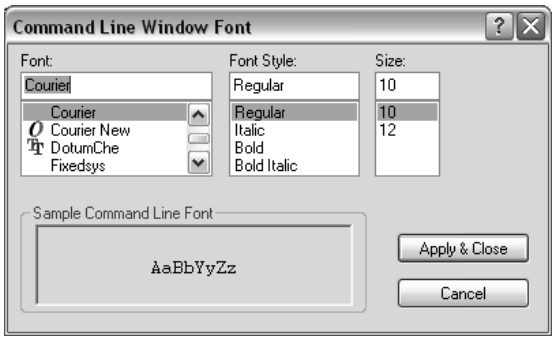
**Restore All Contexts** Restores all interface elements back to their default color settings.

# Command Line Window Font Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, Display tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Sets the font displayed in the command window. The screen menu font is controlled by the Windows system font settings. (If you use the screen menu, you should set the Windows system font setting to a font and font size that fit the screen menu size restrictions.)




**Font** Lists the system fonts available for selection.

**Font Style** Lists the styles available to apply to the current font.

**Size** Lists the font sizes available for the current font.

# Light Glyph Appearance Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, Drafting tab)

Specifies the appearance of the light glyphs.

#### Glyph Preview

Shows the current appearance of the light glyph and updates as you make changes.

**Point** Displays a point light glyph in the preview.

**Spot** Displays a spotlight glyph in the preview.

**Web** Displays a weblight glyph in the preview.

#### Edit Glyph Colors

Displays the Drawing Window Colors dialog box (page 983).

#### Glyph Size

Adjusts the size of the glyph.

## Camera Glyph Appearance Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, Drafting tab)

Specifies the appearance of the camera glyphs.

#### Glyph Preview

Shows the current appearance of the glyph and updates as you make changes.

#### Edit Glyph Colors

Displays the Drawing Window Colors dialog box (page 983).

#### Glyph Size

Changes the size of the glyph in the drawing area.

## Thumbnail Preview Settings Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, Open and Save tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Controls whether thumbnail previews are updated when the drawing is saved.

**Save a Thumbnail Preview Image** Displays an image of the drawing in the Preview area of the Select File dialog box. (*RASTERPREVIEW* system variable)

**Generate Sheet, Sheet View, and Model View Thumbnails** Updates preview images in the Sheet Set Manager. Sheet thumbnails are displayed on the Sheet List tab, sheet view thumbnails on the View List tab, and model space view thumbnails on the Resource Drawings tab. (*UPDATETHUMBNAIL* system variable)

**Performance-Accuracy Slider** Sets the UPDATETHUMBNAIL system variable, which specifies when thumbnails are updated and which thumbnails are updated. When the dialog box is opened, the slider reflects the current setting of UPDATETHUMBNAIL.

**Information Panel** Displays information about the current position of the slider.

## Plot Style Table Settings Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, Plot and Publish tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Specifies settings for plot style tables.

### Default Plot Style Behavior for New Drawings

Controls plot style options. A plot style is a collection of property settings defined in a plot style table and applied when the drawing is plotted. Changing the default plot style setting by using the Options dialog box does not affect the current drawing; it affects only new drawings or drawings created in an earlier release of AutoCAD that have never been saved in AutoCAD 2000 or later format.

The default plot style setting is Use Color Dependent Plot Styles. The Plot Style Control on the Properties toolbar is unavailable by default. The Plot Style Control is available when you select the Use Named Plot Styles option and open a new drawing. (*PSTYLEPOLICY* system variable)

**Use Color Dependent Plot Styles** Uses color-dependent plot styles in both new drawings and drawings created in AutoCAD Release 14 or earlier. Color-dependent plot styles use the numbers from the AutoCAD Color Index to create a plot style table with a *.ctb* file extension. Each color is defined by a name or number ranging from 1 to 255. You can assign each color number to a different pen on a pen plotter to achieve different property settings in the plotted drawing. If this option is selected, a plot style is created for each color setting. You can also select this option by setting the PSTYLEPOLICY system variable to 1.

This setting is saved with the drawing. Once a drawing is saved with Use Color Dependent Plot Styles as the default, you can convert it to Use Named Plot Styles using the *CONVERTCTB* and *CONVERTPSTYLES* commands.

**Use Named Plot Styles** Uses named plot styles in both new drawings and drawings created in earlier versions of AutoCAD. The drawing is plotted according to the property settings that you specify in the plot style definition. The plot style is defined in the plot style table attached to the layout. Named plot style tables are files with the file extension *.stb*. You can also select this option by setting the PSTYLEPOLICY system variable to 0.

This setting is saved with the drawing. Once a drawing is saved with Use Named Plot Styles as the default, you can convert it to Use Color Dependent Plot Styles using the CONVERTCTB and CONVERTPSTYLES commands. However, once you use CONVERTPSTYLES to convert a drawing from using a named plot style table to a color plot style table, you cannot use CONVERTPSTYLES to convert it back to using a named plot style table.

### Current Plot Style Table Settings

Specifies the current plot style table settings that are applied to new drawings.

**Default Plot Style Table** Specifies the default plot style table to attach to new drawings. A plot style table is a file with a *.ctb* or an *.stb* extension that includes and defines plot styles. If you are using color-dependent plot styles, this option lists all color dependent plot style tables found in the search path as well as the value of None. If you are using named plot styles, this option lists all named plot styles tables.

**Default Plot Style for Layer 0** Sets the default plot style for Layer 0 for new drawings or drawings created with earlier releases of AutoCAD that have never been saved in AutoCAD 2000 or later format. The list displays the default style, Normal, and any plot styles defined in the currently loaded plot style table. (*DEFLPLSTYLE* system variable)

**Default Plot Style for Objects** Sets the default plot style that is assigned when you create new objects. The list displays BYLAYER, BYBLOCK, and Normal styles, and any plot styles defined in the currently loaded plot style table. (*DEFPLSTYLE* system variable)

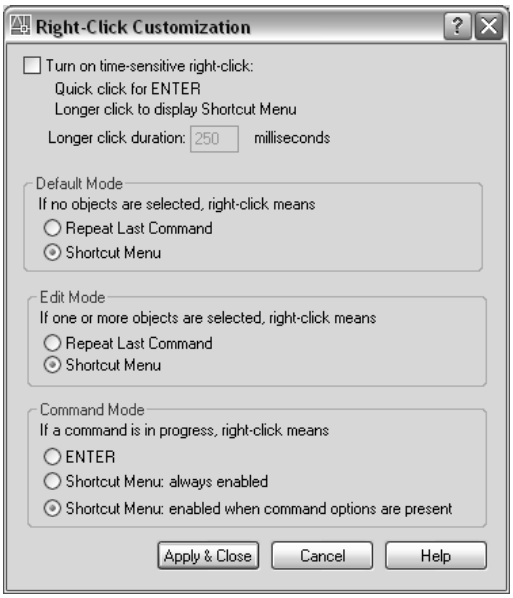
**Add or Edit Plot Style Tables** Displays the Plot Style Manager (page 1372) (a Windows Explorer window), where you can create or edit plot style tables.

# Right-Click Customization Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, User Preferences tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Controls whether right-clicking in the drawing area displays a shortcut menu or has the same effect as pressing ENTER. If you are accustomed to using the right-click method for ENTER while a command is active, you might prefer to disable Command shortcut menus from this dialog box. You can also control shortcut menus by using the *SHORTCUTMENU* system variable.




### Turn on Time-Sensitive Right-Click

Controls right-click behavior. A quick click is the same as pressing ENTER. A longer click displays a shortcut menu. You can set the duration of the longer click in milliseconds.

### Default Mode

Determines what happens when you right-click in the drawing area when no objects are selected and no commands are in progress.

**Repeat Last Command** Disables the Default shortcut menu. As a result, right-clicking in the drawing area when no objects are selected and no commands are in progress is the same as pressing ENTER, which repeats the last issued command.

**Shortcut Menu** Enables the Default shortcut menu.

### Edit Mode

Determines what happens when you right-click in the drawing area when one or more objects are selected and no commands are in progress.

**Repeat Last Command** Disables the Edit shortcut menu. As a result, right-clicking in the drawing area when one or more objects are selected and

no commands are in progress is the same as pressing ENTER, which repeats the last issued command.

**Shortcut Menu** Enables the Edit shortcut menu.

#### Command Mode

Determines what happens when you right-click in the drawing area when a command is in progress.

**ENTER** Disables the Command shortcut menu. As a result, right-clicking in the drawing area when a command is in pro-gress is the same as pressing ENTER.

**Shortcut Menu: Always Enabled** Enables the Command shortcut menu.

**Shortcut Menu: Enabled When Command Options Are Present** Enables the Command shortcut menu only when options are currently available at the command prompt. At the command prompt, options are enclosed in square brackets. If no options are available, right-clicking is the same as pressing ENTER.

## Field Update Settings Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, User Preferences tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Controls how fields are updated. (*FIELDEVAL* system variable)

**Open** Automatically updates fields when the file is opened.

**Save** Automatically updates fields when the file is saved.

**Plot** Automatically updates fields when the file is plotted.

**eTransmit** Automatically updates fields when the file is sent using *ETRANSMIT*.

**Regen** Automatically updates fields when the file is regenerated.

# Visual Effect Settings Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, Selection tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and click Options.
**options**

Controls the appearance of selection previewing.

### Selection Preview Effect

Controls the appearance of objects during selection preview. (*PREVIEWEFFECT* system variable)

**Preview Area** Displays the effect of the current settings.

**Dash** Displays dashed lines when the pickbox cursor rolls over an object. This selection previewing indicates that the object would be selected if you clicked. Dashed lines are the default display for selected objects. (*PREVIEWEFFECT* system variable)

**Thicken** Displays thickened lines when the pickbox cursor rolls over an object. This selection previewing indicates that the object would be selected if you clicked. (*PREVIEWEFFECT* system variable)

**Both** Displays thickened, dashed lines when the pickbox cursor rolls over an object. This selection previewing indicates that the object would be selected if you clicked. (*PREVIEWEFFECT* system variable)

**Advanced Options** Displays the Advanced Preview Options dialog box (page 993).

### Selection Area Effect

Controls the appearance of selection areas during selection preview.

**Preview Area** Displays the effect of the current settings.

**Indicate Selection Area** During window or crossing selection indicates the selection area with a different background color. (*SELECTIONAREA* system variable)

**Window Selection Color** Controls the background for window selection areas. (*WINDOWAREACOLOR* system variable)

**Crossing Selection Color** Controls the background for crossing selection areas. (*CROSSINGAREACOLOR* system variable)

**Selection Area Opacity** Controls transparency of the background for window selection areas. (*SELECTIONAREAOPACITY* system variable)

# Advanced Preview Options Dialog Box

## Quick Reference

Tools ➤ Options (Options dialog box, Selection tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and click Options.
**options**

Excludes objects from selection previewing. (*PREVIEWFILTER* system variable)

### Exclude Objects on Locked Layers

Does not display selection previewing for objects on locked layers. This option is on by default. (PREVIEWFILTER system variable)

### Exclude

Turns off selection previewing for the selected object types. (PREVIEWFILTER system variable)

**Xrefs** Excludes objects in xrefs from selection previewing. This option is on by default. (PREVIEWFILTER system variable)

**Tables** Excludes tables from selection previewing. This option is on by default. (PREVIEWFILTER system variable)

**Groups** Excludes objects in groups from selection previewing. This option is off by default. (PREVIEWFILTER system variable)

**Multiline Text** Excludes multiline text objects from selection previewing. This option is off by default. (PREVIEWFILTER system variable)

**Hatches** Excludes hatch objects from selection previewing. This option is off by default. (PREVIEWFILTER system variable)

## Add Profile Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, Profiles tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Creates a new profile from an existing one. The new profile is displayed in the Profile list box on the Profiles tab in the Options dialog box.

**Profile Name** Assigns a new profile name to the selected profile.

**Description** Assigns a new or updated description for the selected profile.

## Change Profile Dialog Box

### Quick Reference

Tools ➤ Options (Options dialog box, Profiles tab)
Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.
**options**

Renames an existing profile. The new profile name is displayed in the Profile list box on the Profiles tab in the Options dialog box.

**Profile Name** Assigns a new profile name to the selected profile.

**Description** Assigns a new or updated description for the selected profile.

## Transparency Dialog Box

### Quick Reference

Right-click the title bar on the dockable window and click Transparency.

Controls transparency for dockable windows.

**Less/More** Sets the degree of transparency for the window. With the slider at Less, the window is opaque; with the slider at More, the window has maximum transparency.

**Turn Off Window Transparency** When selected, makes all transparent windows opaque. The transparency setting in each window remains unchanged. (*PALETTEOPAQUE* system variable).

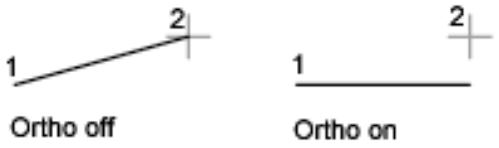
# ORTHO

## Quick Reference

Constrains cursor movement to the horizontal or vertical direction

Status bar ➤ Ortho
**ortho** (or '**ortho** for transparent use)
Enter mode [ON/OFF] <*current*>: *Enter* **on** *or* **off**, *or press* ENTER

In the illustration, a line is drawn using Ortho mode. Point 1 is the first point specified, and point 2 is the position of the cursor when the second point is specified.




Ortho mode is used when you specify an angle or distance by means of two points using a pointing device. In Ortho mode, cursor movement is constrained to the horizontal or vertical direction relative to the UCS.

Horizontal is defined as being parallel to the *X* axis of the UCS and vertical as being parallel to the *Y* axis.

In a 3D view, ORTHO is additionally defined as being parallel to the *Z* axis of the UCS, and the tooltip displays +Z or -Z for the angle depending on the direction along the Z axis.

# OSNAP

## Quick Reference

Sets running object snap modes

Object Snap 
Tools menu ➤ Drafting SettingsAt the Command prompt, enter dsettings.
Press SHIFT while right-clicking in the drawing area and choose Osnap Settings.

Status bar ➤ Osnap 
**osnap** (or '**osnap** for transparent use)

The Object Snaps tab of the Drafting Settings dialog box (page 488) is displayed.

If you enter **-osnap** at the command prompt, the following OSNAP command prompts are displayed.

Current osnap modes: *current*
Enter list of object snap modes (page 996): *Enter names of object snap modes separated with commas, or enter* **none** *or* **off**

### Object Snap Modes

Specify one or more object snap modes by entering the first three characters of the name. If you enter more than one name, separate the names with commas.

| | | |
|---|---|---|
| ENDpoint | CENter | TANgent |
| MIDpoint | NODe | NEArest |
| INTersection | QUAdrant | PARallel |
| EXTension | INSertion | |
| APParent Intersection | PERpendicular | |

For a description of each of these object snap modes, see the Object Snaps tab of the Drafting Settings dialog box (page 488).

The -OSNAP command also presents the following additional options:

**QUIck** Snaps to the first snap point found. Quick must be used in conjunction with other object snap modes.

**NONe** Turns off object snap modes.

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# P Commands

# 16

**In this chapter**

- POINTLIGHT
- POLYGON
- POLYSOLID
- PRESSPULL
- PREVIEW
- PROPERTIES
- PROPERTIESCLOSE
- PSETUPIN
- PSPACE
- PUBLISH
- PUBLISHTOWEB
- PURGE
- PYRAMID

# PAGESETUP

### Quick Reference

Controls the page layout, plotting device, paper size, and other settings for each new layout

File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup. Right-click the Model tab or a layout tab and click Page Setup Manager.
**pagesetup**

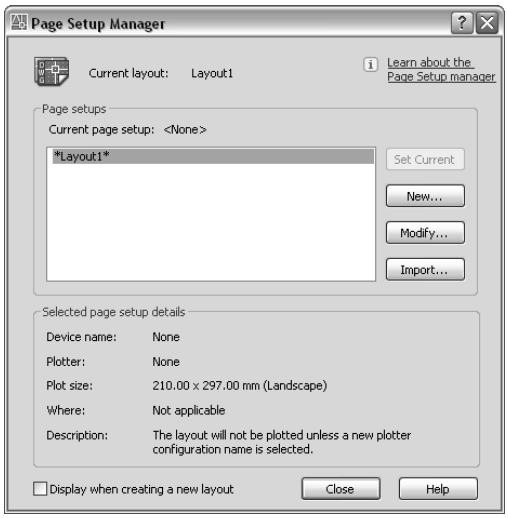
The Page Setup Manager (page 1001) is displayed.

## Page Setup Manager

### Quick Reference

Layouts 
File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup. Right-click the Model tab or a layout tab and click Page Setup Manager.
**pagesetup**

Specifies the page setup for the current layout or drawing sheet. You can also create named page setups, modify existing ones, or import page setups from other drawings.




### Current Layout or Current Sheet Set

Lists the current layout to which the page setup will be applied. If the Page Setup Manager is opened from the Sheet Set Manager, displays the name of the current sheet set. If the Page Setup Manager is opened from a layout, displays the name of the current layout.

**Layout icon** Displayed when the Page Setup Manager is opened from a layout.

**Sheet set icon** Displayed when the Page Setup Manager is opened from the Sheet Set Manager.

### Page Setups

Displays the current page setup, sets a different page setup as current, creates a new page setup, modifies an existing page setup, and imports page setups from other drawings.

**Current Page Setup** Displays the page setup that is applied to the current layout. Displays “Not applicable” if the Page Setup Manager is opened from

the Sheet Set Manager, because you cannot apply a page setup to an entire sheet set after it has been created.

**Page Setup List** Lists the page setups that are available to apply to the current layout or when you publish a sheet set.

If the Page Setup Manager is opened from a layout, the current page setup is selected by default. The list includes the named page setups and layouts that are available in the drawing. Layouts that have a named page setup applied to them are enclosed in asterisks, with the named page setup in parentheses; for example, *Layout 1 (System Scale-to-fit)*. You can double-click a page setup in this list to set it as the current page setup for the current layout.

If the Page Setup Manager is opened from the Sheet Set Manager, only named page setups in the page setup overrides file (a drawing template [*.dwt*] file) that have Plot Area set to Layout or Extents are listed. The first page setup in the list is selected by default. Any of these page setups can be applied temporarily for a *PUBLISH* operation.

The shortcut menu also provides options for removing and renaming page setups.

**Set Current** Sets the selected page setup as the current page setup for the current layout. You cannot set the current layout as the current page setup. Set Current is not available for sheet sets.

**New** Displays the New Page Setup dialog box (page 1004), in which you can enter a name for the new page setup and specify the page setup to use as a starting point.

**Modify** Displays the Page Setup dialog box, (page 1005) in which you can edit the settings for the selected page setup.

**Import** Displays the Select Page Setup From File dialog box (a standard file selection dialog box), in which you can select a drawing format (DWG), DWT, or drawing interchange format (DXF)™ file from which to import one or more page setups. If you select DWT as the file type, the Template folder opens automatically in the Select Page Setup From File dialog box. When you click OK, the Import Page Setups dialog box (page 1013) is displayed.

### Selected Page Setup Details

Displays information about the selected page setup.

**Device Name** Displays the name of the plot device specified in the currently selected page setup.

**Plotter** Displays the type of plot device specified in the currently selected page setup.

**Plot Size** Displays the plot size and orientation specified in the currently selected page setup.

**Where** Displays the physical location of the output device specified in the currently selected page setup.

**Description** Displays descriptive text about the output device specified in the currently selected page setup.

#### Display When Creating a New Layout

Specifies that the Page Setup dialog box is displayed when a new layout tab is selected or a new layout is created.

You can reset this function by turning on the Show Page Setup Dialog for New Layouts option on the Display tab (page 953) of the Options dialog box.

## New Page Setup Dialog Box

### Quick Reference

Layouts

File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup.
Right-click the Model tab or a layout tab and choose Page Setup Manager.
**pagesetup**

Specifies a name for the new page setup, as well as the page setup to use as a starting point.


### New Page Setup Name

Specifies the name for the new page setup.

### Start With

Specifies a page setup to use as a starting point for the new page setup. When you click OK, the Page Setup dialog box (page 1005) is displayed with the settings of the selected page setup, which you can modify as necessary.

If you open the New Page Setup dialog box from the Sheet Set Manager (page 1274), only the named page setups in the page setup overrides file are listed.

**<None>** Specifies that no page setup is used as a starting point. The default settings that are displayed in the Page Setup dialog box (page 1005) can be modified.

**<Default Output Device>** Specifies that the default output device specified in the Options dialog box, Plot and Publish tab, (page 961)is set as the printer in the new page setup.

**<Previous Plot>** Specifies that the new page setup uses the settings specified in the last plot job.

## Page Setup Dialog Box

### Quick Reference

Layouts 

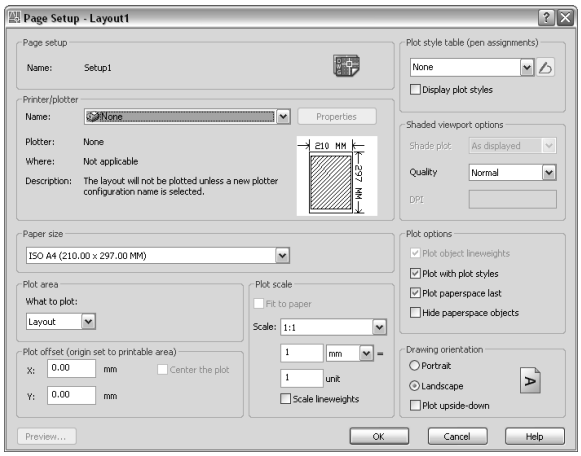
File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup. Right-click the Model tab or a layout tab and choose Page Setup Manager.
**pagesetup**

Specifies page layout and plotting device settings. The Page Setup dialog box is displayed in the following cases:

- When you create a new page setup through the Page Setup Manager (page 1001)
- When you modify an existing page setup through the Page Setup Manager (page 1001)

The page setup settings that you specify are stored with the layout and can be applied to other layouts or imported into other drawings.

The title of the Page Setup dialog box also displays the name of the current layout or sheet set.




### Page Setup

Displays the name of the current page setup.

**Name** Displays the name of the current page setup.

**Icon**

Displays a DWG icon when the Page Setup dialog box is opened from a layout, and displays a sheet set icon when the Page Setup dialog box is opened from the Sheet Set Manager (page 1274).

### Printer/Plotter

Specifies a configured plotting device to use when plotting or publishing layouts or sheets.

**Name** Lists the available PC3 files or system printers from which you can select to plot or publish the current layout or sheet. An icon in front of the device name identifies it as a PC3 file or a system printer.

- *PC3 file icon:* Indicates a PC3 file.

- *System printer icon:* Indicates a system printer.

**Properties** Displays the Plotter Configuration Editor (page 1080) (PC3 editor), in which you can view or modify the current plotter configuration, ports, device, and media settings.

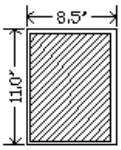
If you make changes to the PC3 file using the Plotter Configuration Editor, the Changes to a Printer Configuration File dialog box (page 1013) is displayed.

**Plotter** Displays the plot device specified in the currently selected page setup.

**Where** Displays the physical location of the output device specified in the currently selected page setup.

**Description** Displays descriptive text about the output device specified in the currently selected page setup. You can edit this text in the Plotter Configuration Editor (page 1080).

**Partial Preview** Shows an accurate representation of the effective plot area relative to the paper size and printable area. The tooltip displays the paper size and printable area.




### Paper Size

Displays standard paper sizes that are available for the selected plotting device. If no plotter is selected, the full standard paper size list is displayed and available for selection.

If the selected plotter doesn't support the layout's selected paper size, a warning is displayed, and you can select the plotter's default paper size or a custom paper size.

A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. For information about this wizard, see "Set Up Plotters and Printers" in the *Driver and Peripheral Guide*. The paper size that you select in the Page Setup dialog box is saved with the layout and overrides the PC3 file settings.

The actual printable area of the page, which is determined by the selected plotting device and paper size, is indicated in the layout by a dashed line.

If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

### Plot Area

Specifies the area of the drawing to be plotted. Under What to Plot, you can select an area of the drawing to be plotted.

**Layout/Limits** When plotting a layout, plots everything within the printable area of the specified paper size, with the origin calculated from 0,0 in the layout.

When plotting from the Model tab, plots the entire drawing area that is defined by the grid limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option.

**Extents** Plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. The drawing may be regenerated to recalculate the extents before plotting.

**Display** Plots the view in the current viewport in the Model tab or in the current paper space view in a layout tab.

**View** Plots a view that was previously saved with the *VIEW* command. You can select a named view from the list. If there are no saved views in the drawing, this option is unavailable.

**Window** Plots any portion of the drawing that you specify. When you specify the two corners of the area to plot, the Window button becomes available.

Click the Window button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values.

Command: Specify first corner: *Specify a point*

Specify other corner: *Specify a point*

### Plot Offset

Specifies an offset of the plot area relative to the lower-left corner of the printable area or to the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab (page 961)). The Plot Offset area of the Page Setup dialog box displays the specified plot offset option in parentheses.

The printable area of a drawing sheet is defined by the selected output device and is represented by a dashed line in a layout. When you change to another output device, the printable area may change.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y offset boxes. The plotter unit values are in inches or millimeters on the paper.

**Center the Plot** Automatically calculates the *X* and *Y* offset values to center the plot on the paper. This option is not available when Plot Area is set to Layout.

**X** Specifies the plot origin in the *X* direction relative to the setting of the Plot Offset Definition option.

**Y** Specifies the plot origin in the *Y* direction relative to the setting of the Plot Offset Definition option.

### Plot Scale

Controls the relative size of drawing units to plotted units. The default scale setting is 1:1 when plotting a layout. The default setting is Fit to Paper when plotting from the Model tab.

**NOTE** If the Layout option is specified in Plot Area, the layout is plotted at 1:1 regardless of the setting specified in Scale.

**Fit to Paper** Scales the plot to fit within the selected paper size and displays the custom scale factor in the Scale, Inch =, and Units boxes.

**Scale** Defines the exact scale for the plot. *Custom* defines a user-defined scale. You can create a custom scale by entering the number of inches (or millimeters) equal to the number of drawing units.

---

**NOTE** You can modify the list of scales with *SCALELISTEDIT*.

---

**Inch(es) =/mm =/Pixel(s) =** Specifies the number of inches, millimeters, or pixels equal to the specified number of units.

**Inch/mm/pixel** Specifies inches or mm for display of units in the Plot dialog box. The default is based on the paper size and changes each time a new paper size is selected. Pixel is available only when a raster output is selected.

**Units** Specifies the number of units equal to the specified number of inches, millimeters, or pixels.

**Scale Lineweights** Scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of plotted objects and are plotted with the linewidth size regardless of the plot scale.

### Plot Style Table (Pen Assignments)

Sets the plot style table, edits the plot style table, or creates a new plot style table.

**Name (Unlabeled)** Displays the plot style table that is assigned to the current Model tab or layout tab and provides a list of the currently available plot style tables.

If you select New, the Add Plot Style Table wizard is displayed, which you can use to create a new plot style table. The wizard that is displayed is determined by whether the current drawing is in color-dependent or named mode.

**Edit** Displays the Plot Style Table Editor (page 1373), in which you can view or modify plot styles for the currently assigned plot style table.

**Display Plot Styles** Controls whether the properties of plot styles assigned to objects are displayed on the screen.

### Shaded Viewport Options

Specifies how shaded and rendered viewports are plotted and determines their resolution levels and dots per inch (dpi).

**Shade Plot** Specifies how views are plotted. To specify this setting for a viewport on a layout tab, select the viewport and then, on the Tools menu, click Properties.

From the Model tab, you can select from the following options:

- *As Displayed:* Plots objects the way they are displayed on the screen.
- *Wireframe:* Plots objects in wireframe regardless of the way they are displayed on the screen.
- *Hidden:* Plots objects with hidden lines removed regardless of the way they are displayed on the screen.
- *3D Hidden:* Plots objects with the 3D Hidden visual style applied regardless of the way the objects are displayed on the screen.
- *3D Wireframe:* Plots objects with the 3D Wireframe visual style applied regardless of the way the objects are displayed on the screen.
- *Conceptual:* Plots objects with the Conceptual visual style applied regardless of the way the objects are displayed on the screen.
- *Realistic:* Plots objects with the Relaistic visual style applied regardless of the way the objects are displayed on the screen.
- *Rendered:* Plots objects as rendered regardless of the way they are displayed on the screen.

**Quality** Specifies the resolution at which shaded and rendered viewports are plotted.

You can select from the following options:

- *Draft*: Sets rendered and shaded model space views to be plotted as wireframe.
- *Preview*: Sets rendered and shaded model space views to be plotted at one quarter of the current device resolution, to a maximum of 150 dpi.
- *Normal*: Sets rendered and shaded model space views to be plotted at one half of the current device resolution, to a maximum of 300 dpi.
- *Presentation*: Sets rendered and shaded model space views to be plotted at the current device resolution, to a maximum of 600 dpi.

- *Maximum*: Sets rendered and shaded model space views to be plotted at the current device resolution with no maximum.
- *Custom*: Sets rendered and shaded model space views to be plotted at the resolution setting that you specify in the DPI box, up to the current device resolution.

**DPI** Specifies the dots per inch for shaded and rendered views, up to the maximum resolution of the current plotting device. This option is available if you select Custom in the Quality box.

### Plot Options

Specifies options for lineweights, plot styles, shaded plots, and the order in which objects are plotted.

**Plot Object Lineweights** Specifies whether lineweights assigned to objects and layers are plotted. This option is unavailable if Plot with Plot Styles is selected.

**Plot with Plot Styles** Specifies whether plot styles applied to objects and layers are plotted. When you select this option, Plot Object Lineweights is automatically selected also.

**Plot Paperspace Last** Plots model space geometry first. Paper space geometry is usually plotted before model space geometry.

**Hide Paperspace Objects** Specifies whether the *HIDE* operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

### Drawing Orientation

Specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation.

**Portrait** Orients and plots the drawing so that the short edge of the paper represents the top of the page.

**Landscape** Orients and plots the drawing so that the long edge of the paper represents the top of the page.

**Plot Upside-Down** Orients and plots the drawing upside-down.

**Icon** Indicates the media orientation of the selected paper and represents the orientation of the drawing on the page as a letter on the paper.

**NOTE** The orientation of plots is also affected by the *PLOTROTMODE* system variable.

**Preview**

Displays the drawing as it will appear when plotted on paper by executing the *PREVIEW* command. To exit the print preview and return to the Page Setup dialog box, press ESC, press ENTER, or right-click and click Exit on the shortcut menu.

# Changes to a Printer Configuration File Dialog Box (Page Setup)

## Quick Reference


Layouts
File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup.
Right-click the Model tab or a layout tab and choose Page Setup Manager.
**pagesetup**

Notifies you that you have made changes to an existing plotter configuration (PC3) file. You can cancel your changes, click OK to overwrite the PC3 file, or specify a new file name for the modified PC3 file.

**Save Changes to the Following File** Specifies the path of the PC3 file that you have changed. To preserve the original PC3 file, specify a new file name.

# Import Page Setups Dialog Box

## Quick Reference


Layouts
File ➤ Page Setup ManagerAt the Command prompt, enter pagesetup.
Right-click the Model tab or a layout tab and choose Page Setup Manager.

**pagesetup**

Displays the page setups in the selected drawing that are available to import.

### Source Drawing

Displays the source drawing that contains the listed page setups that are available to import.

### Page Setups

Lists the page setups that are available to import and their location in the source drawing. Selected page setups are imported into the current drawing when you click OK

When you import page setups into a page setup overrides file for a sheet set, only page setups with Plot Area set to Layout or Extents are listed.

**Name** Specifies the name of the page setup available to import.

**Location** Specifies the location (model or layout) of the page setup in the drawing from which you are importing the page setup.

### Details

Displays information about the selected page setup.

**Device Name** Displays the name of the plot device specified in the currently selected page setup.

**Plotter** Displays the type of plot device specified in the currently selected page setup.

**Plot Size** Displays the plot size and orientation specified in the currently selected page setup.

**Where** Displays the physical location of the output device specified in the currently selected page setup.

**Description** Displays descriptive text about the output device specified in the currently selected page setup.

# PAN

## Quick Reference

Moves the view in the current viewport

Standard 
View ➤ Pan ➤ RealtimeAt the Command prompt, enter pan.
With no objects selected, right-click in the drawing area and choose Pan.
**pan** (or '**pan** for transparent use)

The following prompt is displayed; you can pan the drawing display in real time (page 1015).

Press ESC or ENTER to exit, or right-click to display a shortcut menu (page 1017).

If you enter **-pan** at the command prompt, PAN displays alternative command prompts (page 1016), and you can specify a displacement to pan the drawing display.

# Panning in Real Time

## Quick Reference

Standard
View ➤ Pan ➤ RealtimeAt the Command prompt, enter pan.
With no objects selected, right-click in the drawing area and choose Pan.
**pan** (or '**pan** for transparent use)

The cursor changes to a hand cursor. By holding down the pick button on the pointing device, you lock the cursor to its current location relative to the viewport coordinate system. The drawing display is moved in the same direction as the cursor.

When you reach a logical extent (edge of the drawing space), a bar is displayed on the hand cursor on that edge. Depending on whether the logical extent is

at the top, bottom, or side of the drawing, the bar is either horizontal (top or bottom) or vertical (left or right side).

When you release the pick button, panning stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again to pan the display from that location.

To stop panning at any time, press ENTER or ESC.

# -PAN

## Quick Reference

If you enter **-pan** at the command prompt, the following PAN alternative command prompts are displayed.

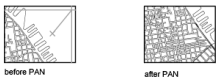
This version of PAN works in two ways. You can specify a single point, indicating the relative displacement of the drawing with respect to the current location, or (more commonly) you can specify two points, in which case the displacement is computed from the first point to the second point.

You cannot use PAN transparently during *VPOINT* or *DVIEW*, or while another *ZOOM*, PAN, or *VIEW* command is in progress.

Specify base point or displacement: *Specify a point (1)*

The point you specify indicates either the amount to move the drawing or the location in the drawing to be moved, depending on how you respond to the next prompt.

Specify second point: *Press* ENTER *or specify a point (2)*




If you press ENTER, the drawing is moved by the amount you specified in the Specify Base Point or Displacement prompt. For example, if you specify **2,2** at the first prompt and press ENTER at the second prompt, the drawing is moved 2 units in the *X* direction and 2 units in the *Y* direction. If you specify

a point at the Specify Second Point prompt, the location of the first point is moved to the location of the second point.

## Pan Shortcut Menu

### Quick Reference

To access the Pan shortcut menu, right-click in the drawing area while PAN is active.

**Exit** Cancels PAN or ZOOM.

**Pan** Switches to PAN.

**Zoom** Switches to ZOOM in real time.

**3D Orbit** Switches to 3DORBIT.

**Zoom Window** Zooms to display an area specified by a rectangular window.

**Zoom Original** Restores the original view.

**Zoom Extents** Zooms to display the drawing extents.

# PARTIALOAD

### Quick Reference

Loads additional geometry into a partially opened drawing

File menu ➤ Partial LoadAt the Command prompt, enter partiaload. **partiaload**

The Partial Load dialog box (page 1018) is displayed. PARTIALOAD can be used only in a partially open drawing. To partially open a drawing, use *OPEN* and choose Partial Open in the Select File dialog box (a standard file selection dialog box (page 931)). Any information that is loaded into the file using PARTIALOAD cannot be unloaded, not even with *UNDO*.

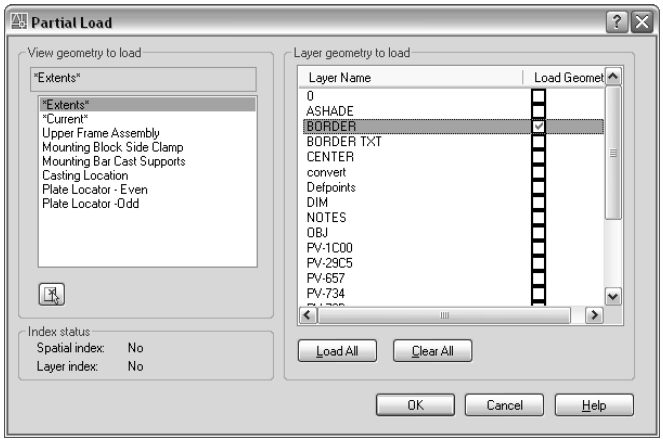
If you enter **-partiaload** at the command prompt, options are displayed at the command prompt (page 1020).

# Partial Load Dialog Box

## Quick Reference

File menu ➤ Partial LoadAt the Command prompt, enter partiaload.
**partialload**

Displays the views and layers available for specifying additional geometry to load into a partially open drawing. You can select and load geometry from a view or a selected area and from layers. You cannot unload any information that is currently loaded in the drawing. If the drawing contains attached xrefs that were not loaded at the time that the drawing was partially opened, you can reload the xrefs by using the External References palette. See *XREF*.




### View Geometry to Load

Displays the selected view and available views in the drawing. Views available for loading include only views defined in model space. You can load paper space geometry by loading the layer on which the paper space geometry is drawn.

**View Name** Displays the currently selected view. Geometry that is common to both the selected view and the layers is loaded into the drawing.

**View List** Displays all the model space views available in the selected drawing file. When a view is selected, only the geometry in the selected view is loaded. The default view is Extents. You can select to load geometry from only one view. Geometry that is common to both the selected view and the layers is

loaded into the drawing. Any information currently loaded into the file cannot be unloaded.

**Pick Window** Prompts you to use the pointing device to specify an area using window selection. The selected area becomes the view to load and is displayed in the View Geometry to Load list as New View.

### Layer Geometry to Load

Displays all the layers available in the selected drawing file. The geometry on selected layers is loaded into the drawing, including both model space and paper space geometry. No layers are selected to load by default. You can load geometry from multiple layers. Geometry that is common to both the selected view and the layers is loaded into the drawing. Any information that is currently loaded into the file cannot be unloaded.

**Layer Name** Displays the layer names in the selected drawing.

**Load Geometry** Specifies whether geometry from a layer is loaded into the drawing or not. If Load Geometry is selected, the geometry from the layer is loaded into the drawing. Any information that is currently loaded into the file cannot be unloaded.

**Load All** Selects Load Geometry for all layers. You can also right-click and choose Load All from the shortcut menu to load geometry from all layers.

**Clear All** Specifies that no geometry be loaded into the drawing (clears the Load Geometry option for all layers). It is recommended that you load geometry from at least one layer into the drawing. If you do not select any layers to load, no geometry is loaded at all, including geometry from the selected view. A warning is issued if you do not select any layers to load into the drawing. You can also right-click and choose Clear All from the shortcut menu to specify that no geometry is loaded from any layer.

### Index Status

Indicates whether the selected drawing file contains a spatial or layer index. *INDEXCTL* controls whether layer and spatial indexes are saved with the drawing file. A spatial index organizes objects based on their location in space. A layer index is a list showing which objects are on which layers.

# -PARTIALOAD

## Quick Reference

When a drawing is partially open and you enter **-partiaload** at the command prompt, the following PARTIALOAD command prompts are displayed.

Specify first corner (page 1020) or [View (page 1020)]: *Specify a point or enter* **v**

### First Corner

Specifies the first corner of a user-defined view.

Specify opposite corner: *Specify a point*

Geometry from the defined view is loaded into the partially open drawing.

Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER

### Layers to Load

Loads geometry from the selected layers into the current drawing, including both model space and paper space geometry. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER
**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

### ?—List Layers

Displays a list of layer names available in the current drawing. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER
**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

### View

Loads the geometry from the specified view into the current drawing.

Enter view to load or [?] <*Extents*>: *Enter a view name, enter* **?**, *or press* ENTER *to load the Extents view*

### View to Load

Loads the geometry from the specified view into the partially open drawing.

Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER
**Layers to Load** Loads geometry from the layer or layers into the current drawing. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

**?—List Layers** Displays a list of layer names available in the current drawing. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

### ?—List Views

Displays a list of model space views available in the selected drawing. Enter a view name.

Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER
**Layers to Load** Loads geometry from one or more layers into the current drawing. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded nto the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

**?—List Layers** A list of layer names available in the current drawing is displayed. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

# PARTIALOPEN

## Quick Reference

Loads geometry and named objects from a selected view or layer into a drawing

**partialopen**

---

**NOTE** It is recommended that you partially open a drawing by using *OPEN* and choosing Partial Open in the Select File dialog box to display the Partial Open dialog box (page 942).

---

When you are running a script and using PARTIALOPEN, *FILEDIA* can be set to 0 or 1. When FILEDIA is set to 0 and you enter **partialopen** or **-partialopen** at the command prompt, the following PARTIALOPEN command prompts are displayed.

Enter name of drawing to open: *Enter a drawing name and press* ENTER

Enter ~ (tilde) at the prompt to ignore FILEDIA and display the Select File dialog box (page 931). In the Select File dialog box, select a file and choose Partial Open to display the Partial Open dialog box (page 942). If you enter a file name at the command prompt, additional prompts are displayed.

Enter view to load (page 1022) or [? (page 1023)] <*Extents*>: *Enter a view name, enter* **?**, *or press* ENTER *to load the Extents view*

### View to Load

Opens the drawing with the geometry from the selected view loaded.

Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER

### Layers to Load

Loads geometry from one or more layers into the current drawing. To load geometry from multiple layers, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

### ?—List Layers

Displays a list of layers available in the selected drawing. Enter one or more layer names. To enter multiple layers, separate the layer names with a comma.

Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER
**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

### ?—List Views

Displays a list of model space views available in the selected drawing. Enter a view name.

Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER

### Layers to Load

Loads geometry from one or more layers into the current drawing. To load geometry from multiple layers, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER
**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

### ?—List Layers

Displays a list of layers available in the selected drawing. Enter one or more layer names. To enter multiple layers, separate the layer names with a comma.

Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER
**Yes** Does not load any external references into the drawing.

**No** Loads all external references into the drawing.

# PASTEASHYPERLINK

## Quick Reference

Inserts data from the Clipboard as a hyperlink

Edit ➤ Paste as Hyperlink
**pasteashyperlink**
Select objects:

The selected objects are inserted in the drawing as hyperlinks.

# PASTEBLOCK

## Quick Reference

Pastes copied objects as a block

Edit ➤ Paste as Block
End any active commands, right-click in the drawing area, and choose Paste as Block.
**pasteblock**
Specify insertion point: *Specify a point*

The objects copied to the Clipboard are pasted into the drawing as a block at the specified insertion point.

# PASTECLIP

## Quick Reference

Inserts data from the Clipboard

Standard 
Edit ➤ PasteAt the Command prompt, enter pasteclip.
End any active commands, right-click in the drawing area, and choose Paste.
**pasteclip**

Specify insertion point:

If the Clipboard contains ASCII text, the text is inserted as a multiline text (mtext) object using the *MTEXT* defaults. A spreadsheet is inserted as a table object.

All other objects, except AutoCAD objects, are inserted as embedded or linked (OLE) objects. You can edit these OLE objects by double-clicking them in the drawing to open the application in which they were created.

**NOTE** You can also use CTRL+V to run PASTECLIP. If the cursor is in the drawing area, PASTECLIP behaves as described. If the cursor is on the command line, text from the Clipboard is pasted at the current prompt.

# PASTEORIG

## Quick Reference

Pastes a copied object in a new drawing using the coordinates from the original drawing

Edit ➤ Paste to Original Coordinates
End any active commands, right-click in the drawing area, and choose Paste to Original Coordinates.
**pasteorig**

The object copied to the Clipboard is pasted into the drawing at the same coordinates used in the original drawing.

PASTEORIG functions only when the Clipboard contains AutoCAD data from a drawing other than the current drawing.

# PASTESPEC

## Quick Reference

Inserts data from the Clipboard and controls the format of the data

Edit ➤ Paste SpecialAt the Command prompt, enter pastespec.
**pastespec**

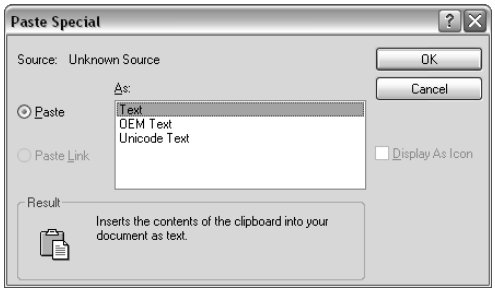
The Paste Special dialog box (page 1026) is displayed.

# Paste Special Dialog Box

## Quick Reference

Edit ➤ Paste SpecialAt the Command prompt, enter pastespec.
**pastespec**

Sets file formats and linking options for pasted files.




**Source** Displays the name of the document that contains the information you copied. It may also show the specific section of the document you copied.

**Paste** Pastes the contents of the Clipboard into the current drawing as an embedded object.

**Paste Link** Pastes the contents of the Clipboard into the current drawing. If the source application supports an OLE or data link, a link is created to the original file.

**As** Displays applicable formats in which you can paste the contents of the Clipboard into the current drawing.

If you select AutoCAD Entities, the metafile graphics in the Clipboard are converted to AutoCAD objects. If the metafile graphics are not converted, the metafile is displayed as an OLE object.

**Display as Icon** Inserts a picture of the application icon instead of the data. To view or edit the data, double-click the icon.

# PCINWIZARD

## Quick Reference

Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout

Tools ➤ Wizards ➤ Import Plot SettingsAt the Command prompt, enter pcinwizard.
**pcinwizard**

PCINWIZARD displays the Import PCP or PC2 Plot Settings wizard. Information that can be imported from PCP or PC2 files includes plot area, rotation, plot offset, plot optimization, plot to file, paper size, plot scale, and pen mapping.

The wizard prompts you for the name of the PCP or PC2 configuration file from which you want to import settings. You can view and modify the plot settings prior to importing them. The imported settings can be applied to the current Model tab or layout tab.

# PEDIT

## Quick Reference

Edits polylines and three-dimensional polygon meshes

Modify II 
Modify ➤ Object ➤ PolylineAt the Command prompt, enter pedit.
Select a polyline to edit, right-click in the drawing area, and choose Polyline Edit.
**pedit**
Select polyline or [Multiple (page 1028)]: *Use an object selection method or enter* **m**

The remaining prompts depend on whether you have selected a 2D polyline (page 1028), a 3D polyline (page 1036), or 3D polygon mesh (page 1039).

If the selected object is a line or an arc, the following prompt is displayed:

Object selected is not a polyline.
Do you want it to turn into one? <Y>: *Enter* **y** *or* **n**, *or press* ENTER

If you enter **y**, the object is converted into a single-segment 2D polyline that you can edit. You can use this operation to join lines and arcs into a polyline. When the *PEDITACCEPT* system variable is set to 1, this prompt is suppressed, and the selected object is automatically converted to a polyline.

### Multiple

Enables selection for more than one object.

### 2D Polyline Selection

If you select a 2D polyline, the following prompt is displayed:

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: *Enter an option or press* ENTER *to end the command*

If the polyline you select is a closed polyline, Open replaces the Close option in the prompt. You can edit a 2D polyline if its normal is parallel to and in the same direction as the *Z* axis of the current UCS.

### Close

Creates the closing segment of the polyline, connecting the last segment with the first. The polyline is considered open unless you close it using the Close option.


### Open

Removes the closing segment of the polyline. The polyline is considered closed unless you open it using the Open option.

### Join

Adds lines, arcs, or polylines to the end of an open polyline and removes the curve fitting from a curve-fit polyline. For objects to join the polyline, their endpoints must touch unless you use the Multiple option at the first PEDIT prompt. In this case, you can join polylines that do not touch if the fuzz distance is set to a value large enough to include the endpoints.

Select objects: *Use an object selection method*

If you previously selected multiple objects using the Multiple option, the following prompt is displayed:

Enter fuzz distance or [Jointype]<0.0000>: *Enter a distance or* **j**

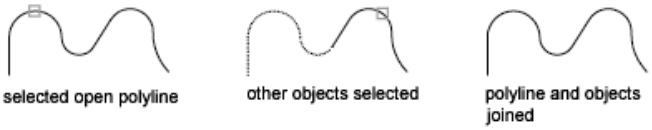
### Jointype

Sets the method of joining selected polylines.

Enter a vertex editing option
Enter join type [Extend/Add/Both}<Extend>: *Enter* **e**, **a**, *or* **b**
**Extend** Joins the selected polylines by extending or trimming the segments to the nearest endpoints.

**Add** Joins the selected polylines by adding a straight segment between the nearest endpoints.

**Both** Joins the selected polylines by extending or trimming if possible. Otherwise joins the selected polylines by adding a straight segment between the nearest endpoints.




### Width

Specifies a new uniform width for the entire polyline.

Specify new width for all segments:

You can use the Width option of the Edit Vertex option to change the starting and ending widths of a segment.


### Edit Vertex

Marks the first vertex of the polyline by drawing an X on the screen. If you have specified a tangent direction for this vertex, an arrow is also drawn in that direction. The following prompt is displayed:

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit]
<*current*>: *Enter an option or press* ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.


### Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline even if the polyline is closed.

### Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline even if the polyline is closed.

### Break

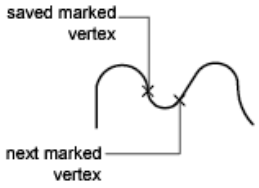
Saves the location of the marked vertex while you move the X marker to any other vertex.

Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER

If one of the specified vertices is at an end of the polyline, the result is one truncated polyline. If both specified vertices are at endpoints of the polyline, or if just one vertex is specified and it is at an endpoint, you cannot use Break.

**Next** Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

**Previous** Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.




**Go** Deletes any segments and vertices between the two vertices you specify and returns to Edit Vertex mode.


**Exit** Exits Break and returns to Edit Vertex mode.

### Insert

Adds a new vertex to the polyline after the marked vertex.

Specify location for new vertex: *Specify a point (1)*


### Move

Moves the marked vertex.

Specify new location for marked vertex: *Specify a point (1)*


### Regen

Regenerates the polyline.

### Straighten

Saves the location of the marked vertex while you move the X marker to any other vertex.

If you want to remove an arc segment that connects two straight segments of a polyline and then extend the straight segments until they intersect, use the *FILLET* command with a fillet radius of 0.

Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER


**Next** Moves the X marker to the next vertex.

**Previous** Moves the X marker to the previous vertex.

**Go** Deletes any segments and vertices between the two vertices you select, replaces them with single straight line segments, and returns to Edit Vertex mode. If you specify only one vertex by entering **go** without moving the X marker, the segment following that vertex is straightened if it is an arc.

**Exit** Exits Straighten and returns to Edit Vertex mode.

### Tangent

Attaches a tangent direction to the marked vertex for use later in curve fitting. The following prompt is displayed:

Specify direction of vertex tangent: *Specify a point or enter an angle*

### Width

Changes the starting and ending widths for the segment that immediately follows the marked vertex.

Specify starting width for next segment <*current*>: *Specify a point, enter a value, or press* ENTER
Specify ending width for next segment <*starting width*>: *Specify a point, enter a value, or press* ENTER

You must regenerate the polyline to display the new width.


### Exit

Exits Edit Vertex mode.

### Fit

Creates an arc-fit polyline, a smooth curve consisting of arcs joining each pair of vertices. The curve passes through all vertices of the polyline and uses any tangent direction you specify.


### Spline

Uses the vertices of the selected polyline as the control points, or frame, of a curve approximating a B-spline. This curve, called a spline-fit polyline, passes through the first and last control points unless the original polyline was closed. The curve is pulled toward the other points but does not necessarily pass through them. The more control points you specify in a particular part of the frame, the more pull they exert on the curve. Quadratic and cubic spline-fit polylines can be generated.


Spline-fit polylines are very different from the curves produced by the Fit option. Fit constructs pairs of arcs that pass through every control point. Both

of these curves are different from true B-splines produced with the *SPLINE* command.

If the original polyline included arc segments, they are straightened when the spline's frame is formed. If the frame has width, the resulting spline tapers smoothly from the width of the first vertex to the width of the last vertex. All intermediate width information is ignored. Once spline-fit, the frame, if displayed, is shown with zero width and CONTINUOUS linetype. Tangent specifications on control point vertices have no effect on spline-fitting.

When a spline-fit curve is fit to a polyline, the spline-fit curve's frame is stored so that it can be recalled by a subsequent decurving. You can turn a spline-fit curve back into its frame polyline by using the PEDIT Decurve option. This option works on fit curves in the same manner as it does on splines.

Spline frames are not usually displayed on the screen. If you want to see them, set the *SPLFRAME* system variable to 1. Next time the drawing is regenerated, both the frame and the spline curve are drawn.


Most editing commands act the same when applied to spline-fit polylines or fit curves.

- *MOVE*, *ERASE*, *COPY*, *MIRROR*, *ROTATE*, and *SCALE* operate on both the spline curve and its frame, whether the frame is visible or not.
- *EXTEND* changes the frame by adding a new vertex where the initial or final line of the frame intersects the boundary geometry.
- *BREAK* and *TRIM* generate a polyline with only the fit spline, which is consistent with fit curves, where the curve fitting is permanent.
- *EXPLODE* deletes the frame and generates lines and arcs to approximate the spline-fit polyline.
- *OFFSET* generates a polyline with only the fit spline, which is consistent with its behavior with fit curves.
- *DIVIDE*, *MEASURE*, and the Object option of *AREA* and *HATCH* see only the fit spline, not the frame.
- *STRETCH* refits the spline to the stretched frame after a spline is stretched.

The Join option of PEDIT decurves the spline and discards the spline information of the original and any added polylines. Once the Join operation is complete, you can fit a new spline to the resulting polyline.

The Edit Vertex options of PEDIT have the following effect:

- The Next and Previous options move the X marker only to points on the frame of the spline, whether visible or not.
- The Break option discards the spline.
- The Insert, Move, Straighten, and Width options automatically refit the spline.
- The Tangent option has no effect on splines.

Object snap uses only the spline-fit curve itself, not the frame. If you want to snap to the frame control points, use PEDIT to recall the polyline frame first.

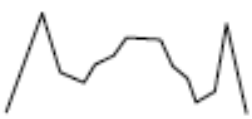
The *SPLINETYPE* system variable controls the type of spline curve approximated. Setting SPLINETYPE to 5 approximates a quadratic B-spline. Setting SPLINETYPE to 6 approximates a cubic B-spline.

quadratic B-spline

cubic B-spline

You can examine or change the fineness or coarseness of the spline approximation with the *SPLINESEGS* system variable, or you can use AutoLISP®. The default value is 8. If you set the value higher, a greater number of line segments are drawn and the approximation to the ideal spline becomes more precise. The generated spline occupies more space in the drawing file and takes longer to generate.

If you set SPLINESEGS to a negative value, the program generates segments using the absolute value of the setting and then applies a fit-type curve to those segments. Fit-type curves use arcs as the approximating segments. Using arcs yields a smoother generated curve when few segments are specified, but the curve can take longer to generate.



SPLINESEGS set to 1

SPLINESEGS set to 10

To change the number of segments used to fit an existing spline, change SPLINESEGS and respline the curve. You do not have to decurve it first.

### Decurve

Removes extra vertices inserted by a fit or spline curve and straightens all segments of the polyline. Retains tangent information assigned to the polyline vertices for use in subsequent fit curve requests. If you edit a spline-fit polyline with a command such as *BREAK* or *TRIM*, you cannot use the Decurve option.

### Ltype Gen

Generates the linetype in a continuous pattern through the vertices of the polyline. When turned off, this option generates the linetype starting and ending with a dash at each vertex. Ltype Gen does not apply to polylines with tapered segments.

Enter polyline linetype generation option [ON/OFF] <*current*>: *Enter* **on** *or* **off**, *or press ENTER*

Ltype Gen set to Off

Ltype Gen set to On

### Undo

Reverses operations as far back as the beginning of the PEDIT session.

### 3D Polyline Selection

If you select a 3D polyline, the following prompt is displayed:

Enter an option [Close/Edit vertex/Spline curve/Decurve/Undo]: *Enter an option or press* ENTER

If the polyline you select is closed, Open replaces the Close option in the prompt.

### Close

Creates the closing segment of the polyline, connecting the last segment with the first. The polyline is considered open unless you close it with Close.

### Open

Removes the closing segment of the polyline. The polyline is considered closed unless you open it with Open.

### Edit Vertex

Performs various editing tasks on one vertex of the polyline and segments that follow it.

Enter a vertex editing option
[Next/Previous/Break/Insert/Move/Regen/Straighten/eXit <*current*>: *Enter an option or press* ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.

### Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

### Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.

### Break

Saves the location of the marked vertex while you move the X marker to any other vertex.

Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER
**Next** Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

**Previous** Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.

**Go** Deletes any segments and vertices between the two vertices you specify and returns to Edit Vertex mode.

**Exit** Exits Break and returns to Edit Vertex mode.

If one of the specified vertices is at an end of the polyline, the polyline is truncated. If both specified vertices are at endpoints of the polyline, or if just one vertex is specified and it is at an endpoint, you cannot use Break mode.

### Insert

Adds a new vertex to the polyline after the marked vertex.

Specify location for new vertex: *Specify a point*

### Move

Moves the marked vertex.

Specify new location for marked vertex: *Specify a point*

### Regen

Regenerates the polyline.

### Straighten

Saves the location of the marked vertex while you move the X marker to any other vertex.

Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER
**Next** Moves the X marker to the next vertex.

**Previous** Moves the X marker to the previous vertex.

**Go** Deletes any segments and vertices between the two vertices you select, replaces them with single straight line segments, and returns to Edit Vertex mode. If you specify only one vertex by entering **go** without moving the X marker, the segment following that vertex is made straight if it is an arc.

**Exit** Exits Straighten and returns to Edit Vertex mode.

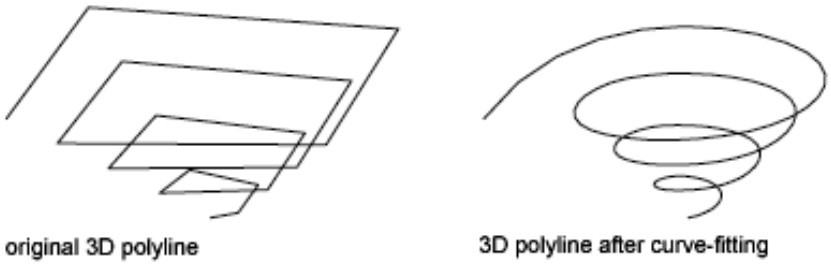
To remove an arc segment that connects two straight segments of a polyline and then to extend the straight segments until they intersect, use the *FILLET* command with a fillet radius of 0.

### Exit

Exits Edit Vertex mode.

### Spline Curve

Fits a 3D B-spline curve to its control points. The *SPLFRAME* system variable controls the accuracy and display of the control points for the 3D B-spline, whose curves can be approximated only by line segments. Negative values for spline segments are ignored.




### Decurve

Removes extra vertices inserted by a fit or spline curve and straightens all segments of the polyline. Retains tangent information assigned to the polyline vertices for use in subsequent fit curve requests. If you edit a spline-fit polyline with a command such as *BREAK* or *TRIM*, you cannot use the Decurve option.

### Undo

Reverses operations as far back as the beginning of the PEDIT session.

### 3D Polygon Mesh Selection

If you select a polygon mesh, the following prompt is displayed:

Enter an option [Edit vertex/Smooth surface/Desmooth/Mclose/Nclose/Undo]:
*Enter an option or press* ENTER *to end the command*

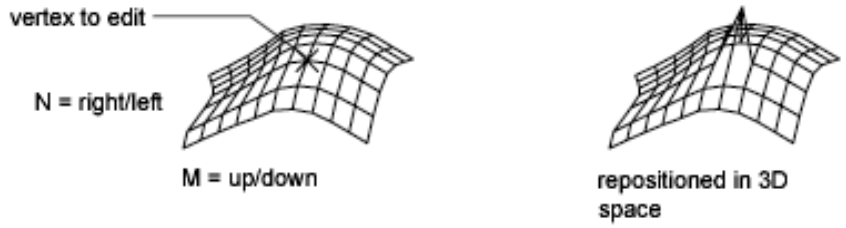
Mclose and Nclose are replaced by Mopen and Nopen if the polygon mesh is currently closed in the *M* or *N* direction.

### Edit Vertex

Edits individual vertices of a polygon mesh that can be seen as a rectangular *M* by *N* array, where *M* and *N* are the dimensions specified in *3DMESH*. The *SURFTAB1* and *SURFTAB2* system variables store *M* and *N* values for *RULESURF*, *TABSURF*, *REVSURF*, and *EDGESURF*.

Enter an option [Next/Previous/Left/Right/Up/Down/Move/REgen/eXit <*current*>: *Enter an option or press* ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.




**Next** Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

**Previous** Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

**Left** Moves the X marker to the previous vertex in the *N* direction. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

**Right** Moves the X marker to the next vertex in the *N* direction. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

**Up** Moves the X marker to the next vertex in the *M* direction. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

**Down** Moves the X marker to the previous vertex in the *M* direction. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

**Move** Repositions the vertex and moves the editing mark.

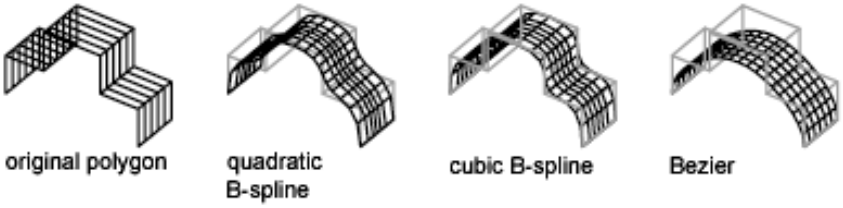
Specify new location for marked vertex: *Specify a point*

**Regen** Regenerates the polygon mesh.

**Exit** Exits Edit Vertex mode.

### Smooth Surface

Fits a smooth surface. The *SURFTYPE* system variable controls the type of surface this option fits. The types of surfaces include quadratic B-spline, cubic B-spline, and Bezier.

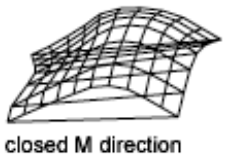



### Desmooth

Restores the original control-point polygon mesh.

### Mclose

Closes the *M*-direction polylines if the polygon mesh is open in the *M* direction.

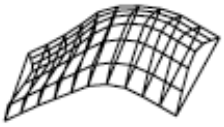



### Mopen

Opens the *M*-direction polylines if the polygon mesh is closed in the *M* direction.

### Nclose

Closes the *N*-direction polylines if the polygon mesh is open in the *N* direction.



closed N direction

#### Nopen

Opens the *N*-direction polylines if the polygon mesh is closed in the *N* direction.

#### Undo

Reverses operations as far back as the beginning of the PEDIT session.

# PFACE

## Quick Reference

Creates a three-dimensional polyface mesh vertex by vertex

**pface**
Specify location for vertex (page 1042) 1: *Specify a point*
Specify location for vertex 2 or <define faces (page 1042)>: *Specify a point or press* ENTER
Specify location for vertex *n* or <define faces>: *Specify a point or press* ENTER

#### Vertex Location

You specify all vertices used in the mesh. The vertex numbers displayed in the prompts are the numbers used to reference each vertex. The prompt is repeated until you press ENTER. If you press ENTER on a blank line, you are prompted for the vertices to be assigned to each face.

#### Define Faces

Enter a vertex number or [Color/Layer]: *Enter a vertex number or enter an option*
**Vertex Number** You define each face by entering vertex numbers for all the vertices of that face. Pressing ENTER after the prompt causes the program to prompt for the vertex numbers of the next face. The mesh is drawn after you have defined the last face and pressed ENTER after the prompt.

To make an edge invisible, you can enter a negative vertex number for the beginning vertex of the edge. The *SPLFRAME* system variable controls the display of invisible edges in polyface meshes. Setting SPLFRAME to a non-zero value displays any phantom faces and all invisible edges of polyface meshes, which you can edit in the same manner as fully visible polyface meshes.

You can create polygons with any number of edges. PFACE automatically breaks them into multiple face objects with the appropriate invisible edges. Faces with one or two vertices behave like point or line objects without the special properties of Point Display modes or linetypes. You can use them to embed wireframe images within a mesh. Use Endpoint object snap to snap to a face composed of one or two vertices. All object snap modes that apply to line objects work with visible edges of polyface meshes. You cannot use *PEDIT* to edit polyface meshes.

**Color** Faces created with PFACE adopt the current layer and color. Unlike polyline vertices, polyface mesh faces can be created with layer and color properties different from their parent object.

New color [Truecolor/COlorbook] <BYLAYER>:*Enter a standard color name or a color number from* **1** *through* **255**, *enter* **t**, *enter* **co**, *or press* ENTER

You can enter a color from the AutoCAD Color Index (a color name or number), a true color, or a color from a color book.

You return to the previous prompt.

**Layer** Faces created with PFACE adopt the current layer and color. Unlike polyline vertices, polyface mesh faces can be created with layer and color properties different from their parent object. Layer visibility behaves normally on faces of a polyface mesh. However, if you create a polyface mesh on a frozen layer, the program does not generate any of its faces, including those on non-frozen layers.

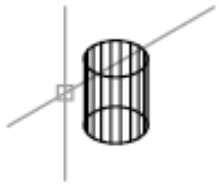
Enter a layer name <0>: *Enter a name or press* ENTER

You return to the previous prompt.

# PLAN

## Quick Reference

Displays the plan view of a specified user coordinate system



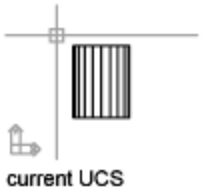
current display

View ➤ 3D Views ➤ Plan ViewDoes not exist in the menus.

**plan**

Enter an option [Current ucs (page 1044)/Ucs (page 1044)/World (page 1044)] <Current>: *Enter an option or press* ENTER

**Current UCS** Regenerates a plan view of the display so that the drawing extents fit in the current viewport of the current UCS.



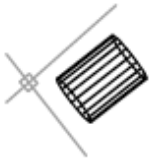
current UCS

**UCS** Changes to a plan view of a previously saved UCS and regenerates the display.

Enter name of UCS or [?]: *Enter a name or enter* **?** *to list all UCSs in the drawing*

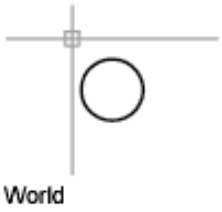
If you enter **?** at the prompt, the following prompt is displayed:

Enter UCS name(s) to list <*>: *Enter a name or enter* ***** *to list all UCSs in the drawing*



UCS

**World** Regenerates a plan view of the display so that the drawing extents fit on the screen of the world coordinate system.




# PLANESURF

## Quick Reference

Creates a planar surface

Modeling 
Draw ➤ Modeling ➤ Planar SurfaceAt the Command prompt, enter Planesurf.
**planesurf**
3D Make panel, Planar Surface

With the PLANESURF command, you can create a planar surface through either of the following methods:

- Select one or more objects that form one or more enclosed areas.
- Specify the opposite corners of a rectangle through the command.

When you specify the corners of the surface through the command, the surface is created parallel to the workplane.

The *SURFU* and *SURFV* system variables control the number of lines displayed on the surface.

Specify first corner or [Object (page 1045)]: *Specify the first point for the planar surface*
Specify other corner: *Specify second point (other corner) for the planar surface*

### Object

Creates a planar or trimmed surface by object selection. You can select one closed object or multiple objects that form a closed area.

Similar to the *REGION* command, valid objects include: line, circle, arc, ellipse, elliptical arc, 2D polyline, planar 3D polyline, and planar spline.

The *DELOBJ* system variable controls whether the object(s) you select are automatically deleted when the surface is created or whether you are prompted to delete the object(s).

Select objects: *Select one or more objects that define the area for the planar surface*

# PLINE

## Quick Reference

Creates two-dimensional polylines

Draw 

Draw ➤ PolylineAt the Command prompt, enter pline.

**pline**

Specify start point: *Specify a point*

Current line-width is <*current*>

Specify next point (page 1046) or [Arc (page 1047)/Close (page 1050)/Halfwidth (page 1051)/Length (page 1051)/Undo (page 1051)/Width (page 1052)]: *Specify a point or enter an option*

**NOTE** At least two points must be specified to use the Close option.

The *PLINEGEN* system variable controls the linetype pattern display around and the smoothness of the vertices of a 2D polyline. Setting PLINEGEN to 1 generates new polylines in a continuous pattern around the vertices of the completed polyline. Setting PLINEGEN to 0 starts and ends the polyline with a dash at each vertex. PLINEGEN does not apply to polylines with tapered segments.



### Next Point

Draws a line segment. The previous prompt is repeated.

### Arc

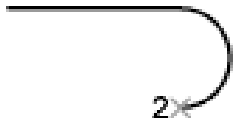
Adds arc segments to the polyline.

Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: *Specify a point (2) or enter an option*

---

**NOTE** For the Center option of the PLINE command, enter **ce**; for the Center object snap, enter **cen** or **center**.

---



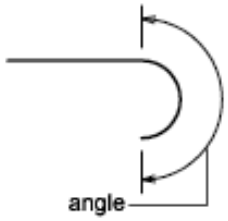

### Endpoint of Arc

Draws an arc segment. The arc segment is tangent to the previous segment of the polyline. The previous prompt is repeated.

### Angle

Specifies the included angle of the arc segment from the start point.

Specify included angle:




Entering a positive number creates counterclockwise arc segments. Entering a negative number creates clockwise arc segments.

Specify endpoint of arc or [Center/Radius]: *Specify a point or enter an option*
**Endpoint of Arc** Specifies the endpoint and draws the arc segment.

**Center** Specifies the center of the arc segment.

Specify center point of arc:

**Radius** Specifies the radius of the arc segment.

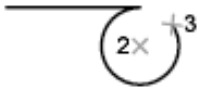
Specify radius of arc: *Specify a distance*

Specify direction of chord for arc <*current*>: *Specify a point or press* ENTER

### Center

Specifies the center of the arc segment.

Specify center point of arc: *Specify a point (2)*
Specify endpoint of arc or [Angle/Length]: *Specify a point (3) or enter an option*




**Endpoint of Arc** Specifies the endpoint and draws the arc segment.

**Angle** Specifies the included angle of the arc segment from the start point.

Specify included angle:

**Length** Specifies the chord length of the arc segment. If the previous segment is an arc, the new arc segment is drawn tangent to the previous arc segment.

Specify length of chord:

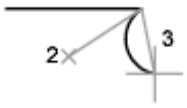
### Close

Draws an arc segment from the last point specified to the starting point, creating a closed polyline. At least two points must be specified to use this option.

### Direction

Specifies a starting direction for the arc segment.

Specify the tangent direction from the start point of arc: *Specify a point (2)*
Specify endpoint of arc: *Specify a point (3)*

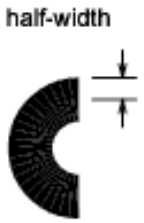


### Halfwidth

Specifies the width from the center of a wide polyline segment to one of its edges.

Specify starting half-width <*current*>: *Enter a value or press* ENTER
Specify ending half-width <*starting width*>: *Enter a value or press* ENTER

The starting half-width becomes the default ending half-width. The ending half-width becomes the uniform half-width for all subsequent segments until you change the half-width again. The starting and ending points of wide line segments are at the center of the line.




Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for nontangent arc segments or very acute angles or when a dot-dash linetype is used.

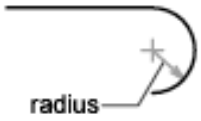
### Line

Exits the Arc option and returns to the initial PLINE command prompts.

### Radius

Specifies the radius of the arc segment.

Specify radius of arc: *Specify a distance*
Specify endpoint of arc or [Angle]: *Specify a point or enter* **a**




**Endpoint of Arc** Specifies the endpoint and draws the arc segment.

**Angle** Specifies the included angle for the arc segment.

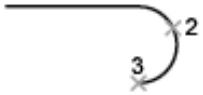
Specify included angle:

Specify direction of chord for arc <*current*>: *Specify an angle or press* ENTER

### Second Pt

Specifies the second point and endpoint of a three-point arc.

Specify second point on arc: *Specify a point (2)*
Specify end point of arc: *Specify a point (3)*



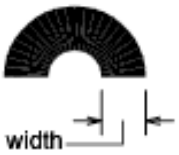

### Undo

Removes the most recent arc segment added to the polyline.

### Width

Specifies the width of the next arc segment.

Specify starting width <*current*>: *Enter a value or press* ENTER
Specify ending width <*starting width*>: *Enter a value or press* ENTER




The starting width becomes the default ending width. The ending width becomes the uniform width for all subsequent segments until you change the width again. The starting and ending points of wide line segments are at the center of the line.

Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for nontangent arc segments, very acute angles, or when a dot-dash linetype is used.

### Close

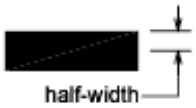
Draws a line segment from the last point specified to the starting point, creating a closed polyline. At least two points must be specified to use this option.

### Halfwidth

Specifies the width from the center of a wide polyline line segment to one of its edges.

Specify starting half-width <*current*>: *Enter a value or press* ENTER
Specify ending half-width <*current*>: *Enter a value or press* ENTER

The starting half-width becomes the default ending half-width. The ending half-width becomes the uniform half-width for all subsequent segments until you change the half-width again. The starting and ending points of wide line segments are at the center of the line.




Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for nontangent arc segments or very acute angles or when a dot-dash linetype is used.

### Length

Draws a line segment of a specified length at the same angle as the previous segment. If the previous segment is an arc, the new line segment is drawn tangent to that arc segment.

Specify length of line: *Specify a distance*

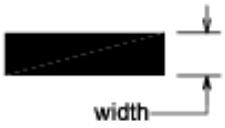
### Undo

Removes the most recent line segment added to the polyline.

### Width

Specifies the width of the next line segment.

Specify starting width <*current*>: *Enter a value or press* ENTER
Specify ending width <*starting width*>: *Enter a value or press* ENTER




The starting width becomes the default ending width. The ending width becomes the uniform width for all subsequent segments until you change the width again. The starting and ending points of wide line segments are at the center of the line.

Typically, the intersections of adjacent wide polyline segments are beveled. No beveling is performed for nontangent arc segments or very acute angles or when a dot-dash linetype is used.

# PLOT

## Quick Reference

Plots a drawing to a plotter, printer, or file

Standard 
File ➤ PlotAt the Command prompt, enter plot.
Right-click the Model tab or a layout tab and click Plot.
**plot**

The Plot dialog box (page 1053) is displayed. Click OK to begin plotting with the current settings.

If you enter **-plot** at the command prompt, options are displayed at the command prompt (page 1065).

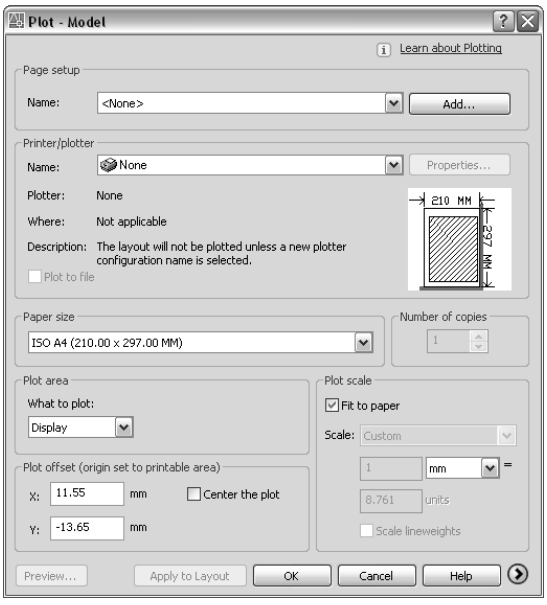
**NOTE** When Texture Compression is turned on, there is a reduction in the quality of the images in the drawing when they are plotted. Texture Compression does not affect viewports that are rendered. To identify if Texture Compression is enabled, enter **3dconfig**, and click Manual Tune. Look at the Hardware Effects List in the Manual Performance Tuning dialog box.

# Plot Dialog Box

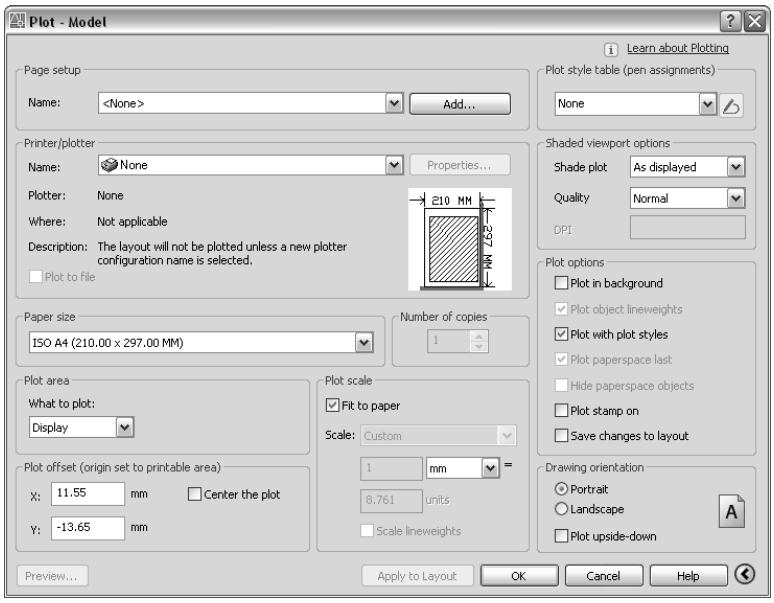
## Quick Reference

Standard
File ➤ PlotAt the Command prompt, enter plot.
Right-click the Model tab or a layout tab and click Plot.
**plot**

Specifies device and media settings, and plots your drawing. The title of the Plot dialog box indicates the name of the current layout.




 You can display more options in the Plot dialog box by clicking the More Options button.




### Page Setup

Displays a list of any named and saved page setups in the drawing. You can base the current page setup on a named page setup saved in the drawing, or you can create a new named page setup based on the current settings in the Plot dialog box by clicking Add.

**Name** Displays the name of the current page setup.

**Add** Displays the Add Page Setup dialog box (page 1062), in which you can save the current settings in the Plot dialog box to a named page setup. You can modify this page setup through the Page Setup Manager.

### Printer/Plotter

Specifies a configured plotting device to use when plotting layouts.

If the selected plotter doesn't support the layout's selected paper size, a warning is displayed and you can select the plotter's default paper size or a custom paper size.

**Name** Lists the available PC3 files or system printers from which you can select to plot the current layout. An icon in front of the device name identifies it as a PC3 file or a system printer.

- *PC3 file icon:* Indicates a PC3 file.

- *System printer icon:* Indicates a system printer.

**Properties** Displays the Plotter Configuration Editor (page 1080) (PC3 editor), in which you can view or modify the current plotter configuration, ports, device, and media settings.

If you make changes to the PC3 file using the Plotter Configuration Editor, the Changes to a Printer Configuration File dialog box (page 1063) is displayed.

**Plotter** Displays the plot device specified in the currently selected page setup.

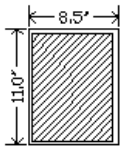
**Where** Displays the physical location of the output device specified in the currently selected page setup.

**Description** Displays descriptive text about the output device specified in the currently selected page setup. You can edit this text in the Plotter Configuration Editor (page 1080).

**Plot to File** Plots output to a file rather than to a plotter or printer. The default location for plot files is specified in the Options dialog box, Plot and Publish tab (page 961), under Default Location for Plot-to-File Operations.

If the Plot to File option is turned on, when you click OK in the Plot dialog box, the Plot to File dialog box (a standard file navigation dialog box) is displayed.

**Partial Preview** Shows an accurate representation of the effective plot area relative to the paper size and printable area. The tooltip displays the paper size and printable area.



### Paper Size

Displays standard paper sizes that are available for the selected plotting device. If no plotter is selected, the full standard paper size list is displayed and available for selection.

If the selected plotter doesn't support the layout's selected paper size, a warning is displayed, and you can select the plotter's default paper size or a custom paper size.

A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. For information about this wizard, see "Set Up Plotters and Printers" in the *Driver and Peripheral Guide*. The paper size that you select in the Page Setup dialog box is saved with the layout and overrides the PC3 file settings.

The actual printable area of the page, which is determined by the selected plotting device and paper size, is indicated in the layout by a dashed line.

If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

### Number of Copies

Specifies the number of copies to plot. This option is not available when you plot to file.

### Plot Area

Specifies the portion of the drawing to be plotted. Under What to Plot, you can select an area of the drawing to be plotted.

**Layout/Limits** When plotting a layout, plots everything within the printable area of the specified paper size, with the origin calculated from 0,0 in the layout.

When plotting from the Model tab, plots the entire drawing area that is defined by the grid limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option.

**Extents** Plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. The drawing may be regenerated to recalculate the extents before plotting.

**Display** Plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.

**View** Plots a view that was previously saved with the *VIEW* command. You can select a named view from the list. If there are no saved views in the drawing, this option is unavailable.

When the View option is selected, a View list is displayed that lists the named views that are saved in the current drawing. You can select a view from this list to plot.

**Window** Plots any portion of the drawing that you specify. When you select Window, the Window button becomes available. Click the Window button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values.

Specify first corner: *Specify a point*

Specify other corner: *Specify a point*

### Plot Offset

Specifies an offset of the plot area relative to the lower-left corner of the printable area or to the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab (page 961)). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

The printable area of a drawing sheet is defined by the selected output device and is represented by a dashed line in a layout. When you change to another output device, the printable area may change.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y offset boxes. The plotter unit values are in inches or millimeters on the paper.

**Center the Plot** Automatically calculates the *X* and *Y* offset values to center the plot on the paper. This option is not available when Plot Area is set to Layout.

**X** Specifies the plot origin in the *X* direction relative to the setting of the Plot Offset Definition option.

**Y** Specifies the plot origin in the *Y* direction relative to the setting of the Plot Offset Definition option.

### Plot Scale

Controls the relative size of drawing units to plotted units. The default scale setting is 1:1 when plotting a layout. The default setting is Fit to Paper when plotting from the Model tab.

**Fit to Paper** Scales the plot to fit within the selected paper size and displays the custom scale factor in the Scale, Inch =, and Units boxes.

**Scale** Defines the exact scale for the plot. *Custom* defines a user-defined scale. You can create a custom scale by entering the number of inches (or millimeters) equal to the number of drawing units.

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**NOTE** You can modify the list of scales with *SCALELISTEDIT*.

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**Inch(es) =/mm =/Pixel(s) =** Specifies the number of inches, millimeters, or pixels equal to the specified number of units.

**Inch/mm/pixel** Specifies inches or mm for display of units in the Plot dialog box. The default is based on the paper size and changes each time a new paper size is selected. Pixel is available only when a raster output is selected.

**Units** Specifies the number of units equal to the specified number of inches, millimeters, or pixels.

**Scale Lineweights** Scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of plotted objects and are plotted with the linewidth size regardless of the plot scale.

### Preview

Displays the drawing as it will appear when plotted on paper by executing the *PREVIEW* command. To exit the print preview and return to the Plot dialog box, press ESC, press ENTER, or right-click and then click Exit on the shortcut menu.

### Apply to Layout

Saves the current Plot dialog box settings to the current layout.

### More Options

Controls display of additional options in the Plot dialog box.

- Plot Style Table (Pen Assignments)
- Shaded Viewport Options
- Plot Options

- Drawing Orientation

### Plot Style Table (Pen Assignments)

Sets the plot style table, edits the plot style table, or creates a new plot style table.

**Name (Unlabeled)** Displays the plot style table that is assigned to the current Model tab or layout tab and provides a list of the currently available plot style tables.

If you select New, the Add Plot Style Table wizard is displayed, which you can use to create a new plot style table. The wizard that is displayed is determined by whether the current drawing is in color-dependent or named mode.

**Edit** Displays the Plot Style Table Editor (page 1373), in which you can view or modify plot styles for the currently assigned plot style table.

### Shaded Viewport Options

Specifies how shaded and rendered viewports are plotted and determines their resolution level and dots per inch (dpi).

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**NOTE** When hardware acceleration is disabled or is enabled, but does not support Shadows, it is possible to plot a drawing that contains shaded viewports with unsupported hardware effects by the graphics card through software emulation. To enable the software emulation of hardware effects that are not supported by your graphics card, enter **3dconfig**, and click Manual Tune. In the Manual Performance Tuning dialog box, click Emulate unsupported hardware effects in software when plotting. The unsupported effects will not appear in the viewport in real-time, but will appear in the hardcopy or electronic file that is created during the plot process.

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**Shade Plot** Specifies how views are plotted. To specify this setting for a viewport on a layout tab, select the viewport and then, on the Tools menu, click Properties.

From the Model tab, you can select from the following options:

- *As Displayed:* Plots objects the way they are displayed on the screen.
- *Wireframe:* Plots objects in wireframe regardless of the way they are displayed on the screen.

- *Hidden:* Plots objects with hidden lines removed regardless of the way the objects are displayed on the screen.
- *3D Hidden:* Plots objects with the 3D Hidden visual style applied regardless of the way the objects are displayed on the screen.
- *3D Wireframe:* Plots objects with the 3D Wireframe visual style applied regardless of the way the objects are displayed on the screen.
- *Conceptual:* Plots objects with the Conceptual visual style applied regardless of the way the objects are displayed on the screen.
- *Realistic:* Plots objects with the Relaistic visual style applied regardless of the way the objects are displayed on the screen.
- *Rendered:* Plots objects as rendered regardless of the way they are displayed on the screen.

**Quality** Specifies the resolution at which shaded and rendered viewports are plotted.

You can select from the following options:

- *Draft*: Sets rendered and shaded model space views to be plotted as wireframe.
- *Preview*: Sets rendered and shaded model space views to be plotted at one quarter of the current device resolution, to a maximum of 150 dpi.
- *Normal*: Sets rendered and shaded model space views to be plotted at one half of the current device resolution, to a maximum of 300 dpi.
- *Presentation*: Sets rendered and shaded model space views to be plotted at the current device resolution, to a maximum of 600 dpi.
- *Maximum*: Sets rendered and shaded model space views to be plotted at the current device resolution with no maximum.
- *Custom*: Sets rendered and shaded model space views to be plotted at the resolution setting that you specify in the DPI box, up to the current device resolution.

**DPI** Specifies the dots per inch for shaded and rendered views, up to the maximum resolution of the current plotting device. This option is available if you select Custom in the Quality box.

### Plot Options

Specifies options for lineweights, plot styles, shaded plots, and the order in which objects are plotted.

**Plot in Background** Specifies that the plot is processed in the background. (*BACKGROUNDPLOT* system variable)

**Plot Object Lineweights** Specifies whether lineweights assigned to objects and layers are plotted. This option is unavailable if Plot with Plot Styles is selected.

**Plot with Plot Styles** Specifies whether plot styles applied to objects and layers are plotted. When you select this option, Plot Object Lineweights is automatically selected also.

**Plot Paperspace Last** Plots model space geometry first. Paper space geometry is usually plotted before model space geometry.

**Hide Paperspace Objects** Specifies whether the *HIDE* operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

**Plot Stamp On** Turns on plot stamping. Places a plot stamp on a specified corner of each drawing and/or logs it to a file.

Plot stamp settings are specified in the Plot Stamp dialog box (page 1070), in which you can specify the information that you want applied to the plot stamp, such as drawing name, date and time, plot scale, and so on. To open the Plot Stamp dialog box, select the Plot Stamp On option, and then click the Plot Stamp Settings button that is displayed to the right of the option.

You can also open the Plot Stamp dialog box by clicking the Plot Stamp Settings button on the Plot and Publish tab (page 961) of the Options dialog box (page 946).

**Plot Stamp Settings button** Displays the Plot Stamp dialog box (page 1070) when the Plot Stamp On option is selected in the Plot dialog box.

**Save Changes to Layout** Saves changes that you make in the Plot dialog box to the layout.

#### Drawing Orientation

Specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation. The paper icon represents the media orientation of the selected paper. The letter icon represents the orientation of the drawing on the page.

**Portrait** Orients and plots the drawing so that the short edge of the paper represents the top of the page.

**Landscape** Orients and plots the drawing so that the long edge of the paper represents the top of the page.

**Plot Upside-Down** Orients and plots the drawing upside down.

**Icon** Indicates the media orientation of the selected paper and represents the orientation of the drawing on the page as a letter on the paper.

**NOTE** Drawing orientation is also affected by the *PLOTROTMODE* system variable.

#### Less Options

Hides the following options in the Plot dialog box:

- Plot Style Table (Pen Assignments)
- Shaded Viewport Options
- Plot Options
- Drawing Orientation

## Add Page Setup Dialog Box

### Quick Reference

Standard
File ➤ PlotAt the Command prompt, enter plot.
Right-click the Model tab or a layout tab and click Plot.
**plot**

Saves the current settings in the Plot dialog box to a named page setup. You can modify this page setup through the Page Setup Manager.

**New Page Setup Name** Specifies a name for the new page setup. You can modify the new named page setup later through the Page Setup Manager.

## Changes to a Printer Configuration File Dialog Box (Plot)

### Quick Reference

Standard
File ➤ PlotAt the Command prompt, enter plot.
Right-click the Model tab or a layout tab and click Plot.
**plot**

Notifies you that you have made changes to an existing plotter configuration (PC3) file. You can cancel your changes, choose OK to overwrite the PC3 file, or specify a new file name for the modified PC3 file.

**Apply Changes for the Current Plot Only** Uses the changes you've made to the PC3 file in the current plot but does not save them in the PC3 file.

**Save Changes to the Following File** Specifies the path of the PC3 file that you have changed. To preserve the original PC3 file, specify a new file name.

## Plot Job Progress Dialog Box

### Quick Reference

File ➤ PlotAt the Command prompt, enter plot.
Right-click the Model tab or a layout tab and choose Plot.
**plot**

The Plot Job Progress dialog box provides information about the status and progress of your plot job.

**Cancel Sheet** Cancels the plot of the sheet currently being processed.

**Cancel Job** Cancels the plot job.

# Update PC3 File with New Printer Dialog Box

## Quick Reference

Displays the device driver name that is stored in the PC3 file. A device can be a plotter or printer. With this dialog box, you can select a valid plotter or printer and update the existing PC3 file; you can also save a modified copy as a new PC3 file.

The dialog box is displayed in the PLOT or PAGESETUP command in a layout configured for a PC3 file that contains an invalid or missing plotter or printer name.

**Device Needed** Displays the device specified in the PC3 file that is configured for the current layout.

**Printer Name** Lists the available plotters, printers, and PC3 files. Select a plotter, printer, or PC3 file to be substituted for the missing printer specified in your PC3 file. Select None if you don't want to make any substitution.

If you select None, the other settings for the current layout remain unchanged. If you select a plotter or printer that uses a different driver than the one specified in the original PC3 file, the following configuration information may change when the PC3 file is saved:

- Plot quality
- Color depth
- Resolution
- Media source
- Media destination
- Media finishing options (such as folding or binding)
- Any custom settings of the previous device or driver

If the plotter or printer you select can't support the paper size specified in the layout, the default paper size is used. A warning is displayed that allows you to cancel this operation and return to the Plot or Page Setup dialog box with the None device selected.

**Device** For each selection you make under Printer Name, the corresponding device driver name is displayed in this column, as follows:

- For a selected plotter or printer, this column displays the device driver name stored in the plotter or printer's PC3 file.
- For a selected PC3 file, this column displays the device driver name stored in the PC3 file.
- For a selection of None, this column displays no device name. No device will be substituted for the missing printer specified in the PC3 file.

**Status Area** Displays information about the selected plotter, printer, or PC3 file.

**Update and Save PC3 File** Displays the name of the PC3 file that you are modifying for the current layout. If you click OK without changing the name of the PC3 file, the existing PC3 file is updated with the new printer name. If you change the name of the PC3 file and click OK, a copy of the PC3 file is saved with the new printer information.

# -PLOT

## Quick Reference

If you enter **-plot** at the command prompt, the following PLOT command prompts are displayed.

Detailed plot configuration [Yes (page 1066)/No (page 1065)] <No>: *Enter* **y** *or* **n** *or press* ENTER

### No

Indicates that you do not want a detailed plot configuration for this plot.

Enter a layout name or [?] <*current*>:
Enter a page setup name < >:
Enter an output device name or [?] <*current*>:
Write the plot to a file [Yes/No] <*current*>:
Enter file name: <*dwgname-layoutname*.plt>:
Save changes to layout [Yes/No] <No>:
Proceed with Plot [Yes/No] <Y>:

For information about these prompts, see the description for Yes.

**Yes**

Specifies detailed page settings for the Model tab or layout tab you are plotting.

Enter a layout name or [?] <*current*>: *Specify the name of the layout tab you want to plot*

Enter an output device name or [?] <*current*>: *Specify the name of the output device to which you want to plot the Model tab or layout tab you selected*

If you enter a new device name without an extension, the program assumes that the device is a PC3 file (Autodesk® HDI plotter configuration file). If no PC3 file is found, the program searches for a Windows system printer with that device name.

Enter paper size or [?] <*current*>: *Specify the paper size to use for the plot or enter* **?** *to view the actual list of paper sizes defined for the plotter driver*

You must specify a paper size exactly as it is defined by the plotter driver.

Enter paper units [Inches/Millimeters] <*current*>:

The Enter Paper Units prompt is not displayed if you are plotting a raster image, such as a BMP or TIFF file, because the size of the plot is assumed to be in pixels.

Enter drawing orientation [Portrait/Landscape] <*current*>:
**Portrait** Orients and plots the drawing so that the short edge of the paper represents the top of the page.

**Landscape** Orients and plots the drawing so that the long edge of the paper represents the top of the page.

Plot upside down [Yes/No] <No>:

Orients and plots the drawing upside down.

Enter plot area [Display/Extents/Limits/Layout/View/Window] <*current*>:
**Display** Plots the view in the current viewport on the Model tab or the current view in the layout, depending on which tab you select to plot.

**Extents** Plots all of the objects in the current viewport, except objects on frozen layers. From a layout tab, plots all the geometry in paper space. The drawing may be regenerated to recalculate the extents before plotting.

If you plot the drawing's extents with a perspective view active and the camera position is within the drawing extents, this option has the same effect as the Display option.

**Limits** Plots the drawing area defined by the grid limits. Available only when the Model tab is selected.

**Layout** Plots everything within the printable area of the specified paper size, with the origin calculated from 0,0 in the layout. Available only when a layout tab is selected.

**View** Plots a view saved previously with the *VIEW* command. You can select a named view from the list provided. If there are no saved views in the drawing, this option is unavailable.

**Window** Plots any portion of the drawing you specify. This option prompts you to specify the corners of the window.

Enter lower left corner of window: *Specify a point*

Enter upper right corner of window: *Specify a point*

Enter plot scale (Plotted Inches = Drawing Units) or [Fit] <*current*>: *Specify the scale of the plot*

**Plotted Inches = Drawing Units** Calculates the plot scale based on the inches or millimeters to drawing units that you specify. You can also enter a real number as a fraction (for example, you can enter **1=2** or **.5**).

**Fit** Calculates the scale to fit the area on the sheet of paper.

The default scale setting is 1:1 when you are plotting a layout, unless you modified and saved the setting. The default setting is Fit when plotting a Model tab.

Enter plot offset (x, y) or [Center] <*current*>: *Specify the plot offset in either the X or Y direction, or enter* **c** *to center the plot on the paper*
Plot with plot styles [Yes/No] <*current*>: *Specify whether to plot using the plot styles applied to objects and defined in the plot style table*

If you specify Yes to plot with plot styles, the following prompt is displayed:

Enter plot style table name or [?] (enter . for none) <*current*>: *Enter a plot style table name,* **?** *to view plot style tables, or* **.** *(period) for none*

All style definitions with different property characteristics are stored in the current plot style table and can be attached to the geometry. This setting replaces pen mapping in earlier versions of the program.

Plot with lineweights [Yes/No] <*current*>:

Scale lineweights with plot scale [Yes/No] <*current*>:

**NOTE** The Scale Lineweights with Plot Scale prompt is displayed only when you plot from a layout tab. Settings for the shaded plotting type are available only when you plot from the Model tab. To control shaded plotting settings of viewports in a layout tab, use the Shadeplot option of the **-vports** command when you create a viewport.

Enter shade plot setting [As displayed/Wireframe/Hidden/Visual styles/Rendered] <*As displayed*>: *Enter a shade plot option*

Specifies how model space views are plotted.

**As Displayed** Specifies that a model space view is plotted the same way it is displayed.

**Wireframe** Specifies that a model space view is plotted in wireframe regardless of display.

**Hidden** Specifies that a model space view is plotted with hidden lines removed regardless of display.

**Visual Styles** Plots a model space view with the specified visual style applied regardless of the current display in the viewport.

If you specify Visual Styles, the following prompt is displayed:

Enter an option [3dwireframe/3dHidden/Realistic/Conceptual/Other] <Realistic>:

- 3D Wireframe. Specifies that a model space view is plotted with the 3D Wireframe visual style applied regardless of display.
- 3D Hidden. Specifies that a model space view is plotted with the 3D Hidden visual style applied regardless of display.
- Realistic. Specifies that a model space view is plotted with the Realistic visual style applied regardless of display.
- Conceptual. Specifies that a model space view is plotted with the Conceptual visual style applied regardless of display.
- Other. Allows you to choose a custom visual style that you've created in the Visual Style manager or you can choose one of the four default visual styles. The following prompt is displayed:
  Enter a visual style name or [?]: *Enter a name or enter* **?** *to display a a list of the visual styles available in the drawing*

**Rendered** Specifies that model space view plots are rendered regardless of display.

Write the plot to a file [Yes/No] <*current*>: *Enter* **y** *if you want to write the plotted drawing to a file, or press* ENTER *to plot to an output device*

If you specify Yes, the following prompt is displayed:

Enter file name: <*dwgname-layoutname.*plt>: *Enter a file name*

Save changes to page setup? Or set shade plot quality? [Yes/No/Quality] <No>:

If you enter **y**, the current settings in the Page Setup dialog box are saved. If you enter **q**, you are prompted for the shaded plotting quality and are given the option of providing a custom dpi. Then you are prompted to save the page setup with the added quality settings.

Enter shade plot quality
[Draft/Preview/Normal/pResentation/Maximum/Custom] <Normal>: *Enter* **c** *if you want to specify a dpi, or to use a preset dpi, specify a different quality option*

Enter custom dpi <150>:

Save changes to page setup [Yes/No]? <No>:

Plot paper space first [Yes/No] <*current*>:

Paper space geometry is usually plotted before model space geometry. If you enter **n**, the model space geometry is plotted first, and paper space geometry is plotted last. This option is available only if you are plotting from a layout tab.

Hide paperspace objects? [Yes/No] <*No*>:

Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab.

Proceed with plot [Yes/No] <Y>:

# PLOTSTAMP

## Quick Reference

Places a plot stamp on a specified corner of each drawing and logs it to a file

**plotstamp**

The Plot Stamp dialog box (page 1070) is displayed.

**NOTE** Plot stamp will always be drawn with pen number 7, or the highest numbered available pen if the plotter doesn't hold seven pens. You must install a suitable pen in that position. If you are using a non-pen (raster) device, color 7 is always used for plot stamping.

If you enter **-plotstamp** at the command prompt, options are displayed at the command prompt (page 1074).

# Plot Stamp Dialog Box

## Quick Reference

### plotstamp

Specifies the information for the plot stamp.

### Plot Stamp Fields

Specifies the drawing information you want applied to the plot stamp. The selected fields are separated by commas and a space.

**Drawing Name** Includes the drawing name and path in the plot stamp information.

**Layout Name** Includes the name of the layout in the plot stamp information.

**Date and Time** Includes the date and time in the plot stamp information.

**NOTE** The date and time format is determined by the Regional Settings dialog box in the Windows Control Panel. Plot stamping uses the short date style for dates.

**Login Name** Includes the Windows login name in the plot stamp information. The Windows login name is contained in the LOGINNAME system variable.

**Device Name** Includes the current plotting device name in the plot stamp information.

**Paper Size** Includes the paper size for the currently configured plotting device in the plot stamp information.

**Plot Scale** Includes the plot scale in the plot stamp information.

### Preview

Provides a visual display of the plot stamp location based on the location and rotation values you have specified in the Advanced Options dialog box. You cannot preview the plot stamp any other way. This is not a preview of the plot stamp contents.

### User Defined Fields

Provides text that can optionally be plotted, logged, or both plotted and logged at plot time. The selected value in each user-defined list will be plotted. For example, you might populate one list with media types or prices and the other with job names. If the user-defined value is set to <none>, then no user-defined information is plotted.

**Add/Edit** Displays the User Defined Fields dialog box (page 1072), where you can add, edit, or delete user-defined fields.

### Plot Stamp Parameter File

Stores plot stamp information in a file with a *.pss* extension. Multiple users can access the same file and stamp their plots based on company standard settings.

Two PSS files are provided, *Mm.pss* and *Inches.pss*, which are located in the *Support* folder. The initial default plot stamp parameter file name is determined by the regional settings of the operating system when the program is installed.

**Path** Specifies the location of the plot stamp parameter file.

**Load** Displays the Plotstamp Parameter File Name dialog box (a standard file selection dialog box) in which you can specify the location of the parameter file you want to use.

**Save As** Saves the current plot stamp settings in a new parameter file.

### Advanced

Displays the Advanced Options dialog box (page 1072), in which you can set the location, text properties, and units of the plot stamp. You can also create a log file, and set its location.

# User Defined Fields Dialog Box

## Quick Reference

**plotstamp**

User-defined fields are created and edited using the User Defined Fields dialog box, which is displayed when you choose the Add/Edit button in the Plot Stamp dialog box.

**Name** Lists the available user-defined fields.

**Add** Adds an editable user-defined field to the bottom of the list.

**Edit** Allows editing of the selected user-defined field.

**Delete** Deletes the selected user-defined field.

# Advanced Options Dialog Box

## Quick Reference

**plotstamp**

Determines the location, text properties, and units of the plot stamp. You can also create a log file and set its location. The dialog box is displayed when you choose the Advanced button in the Plot Stamp dialog box.

### Location and Offset

Determines the plot stamp location, the orientation of the plot stamp, and the offset you want to apply relative to either the printable area or the paper border.

**Location** Indicates the area where you want to place the plot stamp. Selections include Top Left, Bottom Left (default), Bottom Right, and Top Right. The location is relative to the image orientation of the drawing on the page.

**Orientation** Indicates the rotation of the plot stamp in relation to the specified page. The options are Horizontal and Vertical for each of the locations (for example, Top Left Horizontal and Top Left Vertical).

**Stamp Upside Down** Rotates the plot stamp upside down.

**X Offset** Determines the *X* offset value that is calculated from either the corner of the paper or the corner of the printable area, depending on which setting you specify. If you specify Offset Relative to Paper Border, the offset value is calculated so that the plot stamp information fits within the designated paper size. If the offset value positions the plot stamp information beyond the printable area, the plot stamp text is cut off.

**Y Offset** Determines the *Y* offset value that is calculated from either the corner of the paper or the corner of the printable area, depending on which setting you specify. If you specify Offset Relative to Paper Border, the offset value is calculated so that the plot stamp information fits within the designated paper size. If the offset value positions the plot stamp information beyond the printable area, the plot stamp text is cut off.

**Offset Relative to Printable Area** Calculates the offset values that you specify from the corner of the printable area of the paper (not the corner of the paper).

**Offset Relative to Paper Border** Calculates the offset values that you specify from the corner of the paper (not the corner of the printable area of the paper).

### Text Properties

Determines the font, height, and number of lines you want to apply to the plot stamp text.

**Font** Specifies the TrueType font you want to apply to the text used for the plot stamp information.

**Height** Specifies the text height you want to apply to the plot stamp information.

**Single Line Plot Stamp** If selected, places the plot stamp information in a single line of text. The plot stamp information can consist of up to two lines of text, but the placement and offset values you specify must accommodate text wrapping and text height. If the plot stamp contains text that is longer than the printable area, the plot stamp text will be cut off. If this option is cleared, plot stamp text is wrapped after the third field.

### Plot Stamp Units

Specifies the units used to measure *X* offset, *Y* offset, and height. You can define units using inches, millimeters, or pixels.

Two sets of values for the plot stamp size and location are saved in the PSS file: one for the dimensionless file formats and the other for dimensional file formats. If the units you select in the dialog box are dimensional, then the

dimensional values are displayed and modified. To access the dimensionless values, you need to select pixels as the unit of measurement. Regardless of the units selected in the dialog box, the correct set of values is applied at plot time. In other words, changing the unit of measurement does not cause the values to be recalculated.

### Log File Location

Writes the plot stamp information to a log file instead of, or in addition to, stamping the current plot. If plot stamping is turned off, the log file can still be created.

**Create a Log File** Writes the plot stamp information to a log file. The default log file is *plot.log*, and it is located in the main application folder. You can specify a different file name and path. After the initial *plot.log* file is created, the plot stamp information in each succeeding plotted drawing is added to this file. Each drawing's plot stamp information is a single line of text. The plot stamp log file can be placed on a network drive and shared by multiple users. Plot stamp information from each user is appended to the *plot.log* file.

**Log File Name** Specifies the file name for the log file you are creating. Enter a new file name if you do not want to use the default file name, *plot.log*.

**Browse** Lists the currently saved plot stamp log files. You can choose to overwrite an existing plot stamp log file with the currently specified plot stamp information, and then to save this file.

# -PLOTSTAMP

## Quick Reference

If you enter **-plotstamp** at the command prompt, the following PLOTSTAMPcommand prompts are displayed. The settings in the PSS file are displayed as defaults for each prompt.

You can use -PLOTSTAMP as part of a plotting script to modify plot stamp information for a drawing.

Enter an option [On (page 1074)/OFF (page 1074)/Fields (page 1075)/User fields (page 1075)/Log file (page 1075)/LOCation (page 1075)/Text properties (page 1076)/UNits (page 1076)]:
**On** Turns on the plot stamp for the current drawing.

**OFF** Turns off the plot stamp for the current drawing.

**Fields** Specifies the plot stamp field information you want to apply to the current plot stamp.

Stamp drawing name? [Yes/No] <Yes>:

Stamp layout name? [Yes/No] <Yes>:

Stamp date and time? [Yes/No] <Yes>:

Stamp login name? [Yes/No] <Yes>:

Stamp plot device name? [Yes/No] <Yes>:

Stamp paper size? [Yes/No] <Yes>:

Stamp plot scale? [Yes/No] <Yes>:

**User Fields** Specifies the user-defined fields you want to apply to the current plot stamp.

Enter User field 1 <>: *Enter any user-defined field*

Enter User field 2 <>: *Enter any user-defined field*

**Log File** Specifies writing the current plot stamp information to a log file rather than applying this information to the current plotted drawing. The default log file is *plot.log*, unless you specify another file path.

Write plot stamp to log file? [Yes/No] <Yes>:

Enter log file path <plot.log>:

**Location** Determines the location of the plot stamp on the page based on offset, orientation, and relationship to either the printable area or the border of the paper.

Location selections include and are relative to the printable area or the border of the paper, depending on what you specify at the prompt.

Enter stamp location [TL/TR/BL/BR] <BL>:

- *TL:* Top Left
- *TR:* Top Right
- *BL:* Bottom Left
- *BR:* Bottom Right

Text orientation indicates the rotation angle of the plot stamp in relation to the page.

Enter text orientation [Horizontal/Vertical] <Horizontal>:

- *Horizontal:* Plot stamp will be horizontal relative to the page.
- *Vertical:* Plot stamp will be vertical relative to the page.

Stamp upside-down [Yes/No] <No>:

Specify plot stamp offset <0.1000,0.1000>:

Specifying an offset relative to the paper border calculates the offset values that you specify from the corner of the paper. Specifying an offset relative to the printable area calculates the offset values that you specify from the corner of the printable area.

Specify offset relative to [paper Border/printable Area] <printable Area>:

**Text Properties** Determines the font name and text height for the current plot stamp text. You can also specify to place the text on one line or to wrap the text to two lines. The placement and offset values you specify for this plot stamp must accommodate the text wrapping and the text height.

Enter font name <>: *Enter a font name*

Enter text height <0.1500>: *Enter a value*

Place plot stamp on single line? [Yes/No] <No >:

**Units** Specifies the units used to measure *X* offset, *Y* offset, and height. You can define units using inches, millimeters, or pixels.

Enter measurement units [Inches/Millimeters/Pixels] <Inches>:

# PLOTSTYLE

## Quick Reference

Sets the current plot style for new objects or assigns a plot style to selected objects

**plotstyle**

To use the PLOTSTYLE command, the drawing must be configured to use named plot styles.

- If a drawing is in Color-Dependent mode, you can convert it to use named plot styles using the *CONVERTPSTYLES* command.
- To configure a new drawing to use named plot styles, set *PSTYLEPOLICY* to 0 before creating the new drawing.

If no selection set exists, the Current Plot Style dialog box (page 1077) is displayed. If a selection set exists, the Select Plot Style dialog box (page 1078) is displayed

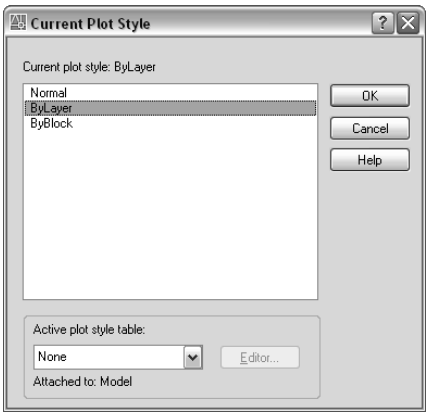
If you enter **-plotstyle** at the command prompt, a options are displayed at the command prompt (page 1079).

# Current Plot Style Dialog Box

## Quick Reference

**plotstyle**

Specifies the plot style for new objects.




**Current Plot Style** Displays the current plot style.

**Plot Style List** Displays the available plot styles that can be assigned to an object, including the default plot style, NORMAL. A plot style is a collection of overrides for color, dithering, gray scale, pen assignments, screening, linetype, lineweight, end styles, join styles, and fill styles.

**Active Plot Style Table** Sets the plot style table attached to the current layout. A plot style table is a collection of plot style definitions.

**Editor** Displays the Plot Style Table Editor (page 1373), in which you modify the plot styles in the plot style table.

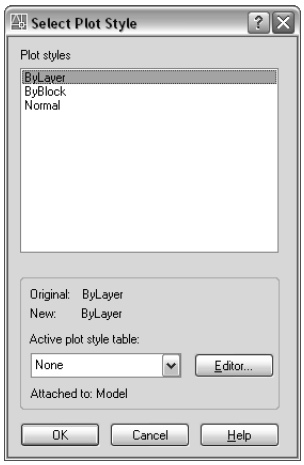
**Attached To** Displays the tab (Model tab or layout tab) that the plot style table is attached to.

# Select Plot Style Dialog Box

## Quick Reference

### plotstyle

Specifies the plot style to be assigned.




**Plot Styles** Displays the available plot styles, including the default plot style, NORMAL. A plot style is a collection of property settings used in plotting.

**Original** Displays the originally assigned plot style.

**New** Displays the new plot style to be assigned.

**Active Plot Style Table** Sets the plot style table attached to the current drawing. A plot style table defines plot styles.

**Editor** Displays the Plot Style Table Editor (page 1373).

**Attached To** Displays the tab (Model tab or layout tab) that the plot style table is attached to.

# -PLOTSTYLE

## Quick Reference

If you enter **-plotstyle** at the command prompt, the following PLOTSTYLE command prompts are displayed.

**-plotstyle**
Current plot style is "*current*"
Enter an option [?/Current]:
**?—List Plot Styles** Lists the plot styles in the attached plot style table.

**Current** Specifies the plot style to use for new objects.

# PLOTTERMANAGER

## Quick Reference

Displays the Plotter Manager, where you can add or edit a plotter configuration

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

The Plotter Manager is displayed, where you can perform the following tasks:

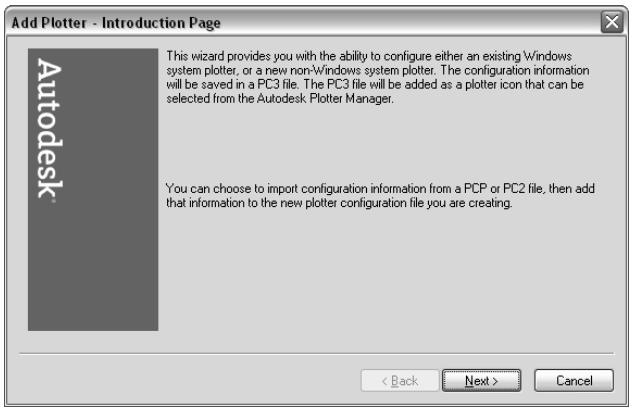
- Double-click the Add-a-Plotter wizard (page 1079) to add and configure plotters and printers.
- Double-click a plotter configuration (PC3 file) to start the Plotter Configuration Editor (page 1080).

# Add-a-Plotter Wizard

## Quick Reference

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

Adds new plotters and printers. The wizard produces a PC3 file that you can edit in the Plotter Configuration Editor (page 1080). Your configurations are stored in the *Plotters* folder.




# Plotter Configuration Editor

## Quick Reference

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

Modifies a PC3 file's plotter port connections and output settings, including media, graphics, physical pen configuration, custom properties, initialization strings, calibration, and user-defined paper sizes. You can drag these configuration options from one PC3 file to another.

You can start the Plotter Configuration Editor with any of the following methods:

- Double-click a PC3 file in Microsoft® Windows® Explorer or right-click the PC3 file and click Open. (By default, PC3 files are stored in the *Plotters* folder. To find the location of your plotter files, on the Tools menu, click Options. In the Options dialog box, Files tab, click the plus sign to the left of Printer Support File Path. Click the plus sign to the left of the Printer Configuration Search Path file. Under Printer Configuration Search Path, click the path name to view the location of your plotter files.)
- Choose Edit Plotter Configuration from within the Add-a-Plotter wizard.

- Choose Properties in the Page Setup dialog box.
- Choose Properties in the Plot dialog box.

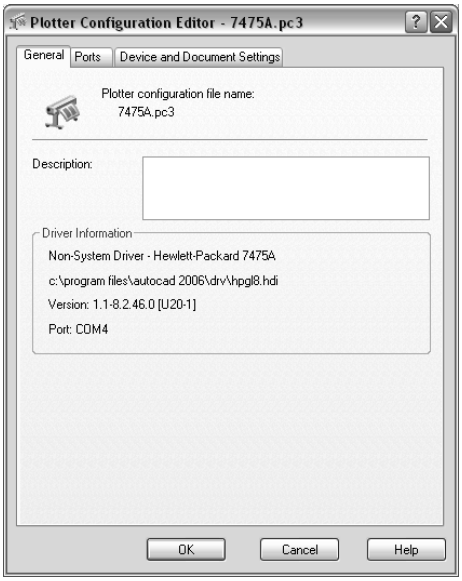
Depending on your configured plotting device, additional options may be available on the Device and Document Settings tab. For example, when you configure a nonsystem plotter, you can modify the pen characteristics.

- General (page 1081)
- Ports (page 1082)
- Device and Document Settings (page 1084)

# General Tab (Plotter Configuration Editor)

## Quick Reference

Contains basic information about the plotter configuration (PC3) file. You can add or modify the information in the Description area. The remainder of the tab is read-only.



**Plotter Configuration File Name** Displays the file name you assigned in the Add-a-Plotter wizard.

**Description** Displays information you want to include about the plotter.

**Driver Information** Displays the following information:

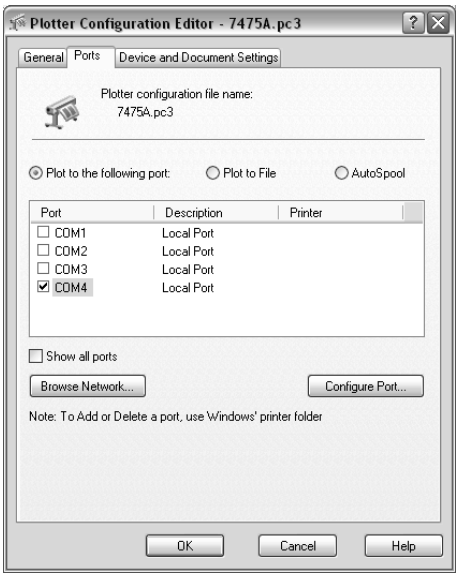
- Plotter driver type (system or nonsystem), name, model, and location
- HDI driver file version number (AutoCAD specialized driver files)
- UNC name of the network server (if the plotter is connected to a network server)
- I/O port (if the plotter is connected locally)
- Name of the system printer (if the configured plotter is the system printer)
- PMP (Plot Model Parameters) file name and location (if a PMP file is attached to the PC3 file)

## Ports Tab (Plotter Configuration Editor)

### Quick Reference

Changes communication settings between the configured plotter and your computer or network system. You can specify whether you want to plot through a port, plot to a file, or use AutoSpool. See "Use AutoSpool" in the *Driver and Peripheral Guide*.

If you plot through a parallel port, you can specify the timeout value. If you plot through a serial port, you can change the baud rate, protocol, flow control, and input and output timeout values.




**Plot to the Following Port** Sends the drawing to the plotter through the selected port.

**Plot to File** Sends the drawing to the file specified in the Plot dialog box.

**AutoSpool** Uses the AutoSpool utility to plot the drawing. AutoSpool is specified on the Files tab in the Options dialog box.

**Port List** Displays a list and description of available ports, both local and network. If the port is a network port, the name of the network printer is also displayed.

**Show All Ports** Shows all available ports on the computer regardless of the ports available on the plotter.

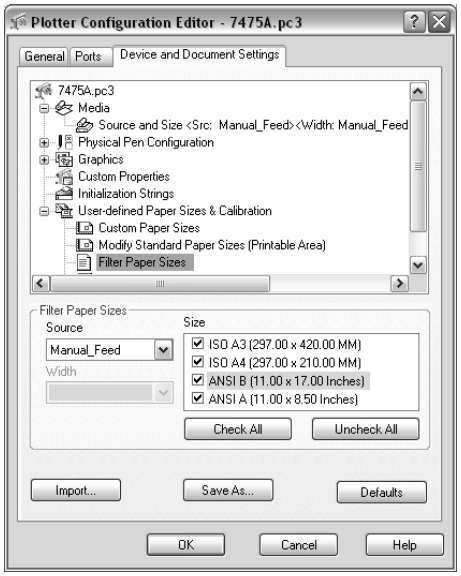
**Browse Network** Displays the network choices so you can connect to another instance of a nonsystem plotter.

**Configure Port** Displays either the Configure LPT Port dialog box (page 1092) or the Settings for COM Port dialog box (page 1092).

# Device and Document Settings Tab (Plotter Configuration Editor)

## Quick Reference

Controls many of the settings in the PC3 file. Click the icon for any of the nodes to view and change the specific settings. When you change a setting, your changes appear in angle brackets (< >) next to the setting name. A check mark is also displayed over the icon of the node with a changed value.




**NOTE** Only the settings available for the configured device are displayed in the tree view. In addition, you might not be able to edit some settings if the device handles the setting through Custom Properties or doesn't support the capability.

### Media Node

Specifies a paper source, size, type, and destination. Available settings depend on the supported features of your configured plotter. For Windows system printers, you must configure the media settings using the Custom Properties node.

**Source and Size** Specifies the paper source and size.

- *Source:* Specifies the paper source; for example, sheet-fed or roll-fed. If you specify a tray, you can select the type of tray.
- *Width:* Specifies the width of the paper roll for a roll-fed source.
- *Automatic:* Enables the printer to specify the appropriate paper source.
- *Size:* Displays a list of available paper sources and both standard and custom paper sizes.
- *Printable Bounds:* Displays the print boundaries.

**Media Type** Displays a list of the media types supported by the plotter configuration.

**Duplex Printing** Determines double-sided printing and binding margin. Binding margin options are available only for plotters that support duplex printing.

- *None:* Indicates no double-sided printing.
- *Short Side:* Places the binding margin on the short side of the paper.
- *Long Side:* Places the binding margin on the long side of the paper.

**Media Destination** Displays a list of available media destinations for the configured plotter, such as collating, cutting, and stapling. These options are available only for plotters that support this function.

### Physical Pen Configuration Node (for Pen Plotters Only)

Controls the specific pens in the pen plotter. The lower pane of the Device and Document Settings tab displays a table used to describe the color, width, and speed of each pen in the plotter.

---

**NOTE** The physical pen information cannot be detected automatically; you must provide this information for your pen plotter under Physical Pen Characteristics.

---

**Pen Configuration** Specifies settings for pen plotters.

- *Prompt for Pen Swapping:* Use more than one pen while plotting on a single-pen plotter. Under Physical Pen Characteristics, you can specify settings for as many pens as you want. You are prompted when you need to change pens.

- *Area Fill Correction:* Enables the program to compensate for pen widths when plotting filled areas and wide polylines. Each polygon is shrunk by half of the width of the pen used to draw it. This option prevents the plotter from oversizing the filled area when using a wide pen and ensures precision. Use this option if the plot must be accurate to half the pen width, such as in printed circuit artwork.
- *Pen Optimization Level:* Reduces plot time and increases the efficiency of the pens by optimizing the pen motion. For example, you can prevent pens from retracing duplicate lines. If your drawing uses many colors or widths, you can reduce the time needed to change pens by selecting Adds Pen Sorting. Every object that uses a particular pen will be plotted before switching to another pen. Each method in the list includes the optimization methods preceding it in the list (except for No Optimization).

**Physical Pen Characteristics** To plot your drawing correctly on a pen plotter, you need to provide information about the pens in your plotter. For each pen in your plotter, specify a color and width. To optimize pen performance, you can specify a speed.

---

**NOTE** This information is required; the physical pen information cannot be detected automatically.

---

- *Color:* Specifies the colors of the pens in your plotter. The pen color that closely matches the object's color is selected. You can assign colors to your pens that correspond to each object's color. Use the Color list to select one of the following settings: No Pen, Red, Yellow, Green, Cyan, Blue, Magenta, Black, or Other. If you choose Other, the Select Color dialog box is displayed, providing access to the full color palette. Use No Pen to indicate that there is no pen in a particular position.
- *Speed:* Adjusts pen speed on a pen-by-pen basis. This feature is useful, for example, for slowing down pens that are skipping. Each pen manufacturer recommends a pen speed for each type of media. For best results, use those values. You can specify a pen speed in millimeters or inches per second.
- *Width:* Specifies the width of your pens so the program can determine if multiple pen strokes are needed to draw wide lines. You can specify the pen width in inches or millimeters. Be sure to select the pen width to match the actual width of the pen. The list provides a set of common pen widths.

Specifying pen characteristics here doesn't replace the pen tables you might have imported from PCP, PC2, or CFG files from AutoCAD Release 14,

AutoCAD LT® 98, or earlier releases. For more information about importing these settings, see Change Plot Style Settings in the *User's Guide*.

### Graphics Node

Specifies settings for printing vector graphics, raster graphics, and TrueType text. Depending on the capabilities of the plotter, you can modify color depth, resolution, and dithering. You can select either color or monochrome output for vector drawings. When printing raster images on a plotter with limited memory, you can improve performance by making some changes to the quality of the printed output. If you use a nonsystem plotter that supports varying amounts of installed RAM, you can provide that information to improve performance.

**Installed Memory** Provides the program with the amount of total memory (RAM) installed on a nonsystem plotter. This option is only available for nonWindows system printers that accept optional memory. If your plotter has extra memory, specify the total amount of memory.

- *Total Installed Memory:* Specifies the total installed memory in megabytes. The plotter driver uses the memory information to determine whether banding or image quality degradation is required to prevent the system from running out of memory.

**Vector Graphics** Provides options for specifying the color depth, resolution, and dithering of vector output. Some of the Vector Graphics options are closely interrelated; changing an option can affect other available options.

- *Color Depth:* Displays a list for choosing color depth for the configured plotter. The color depth options change as you change the resolution and dithering values. More color depth uses more memory and takes longer to plot. You can specify either color or monochrome output.
- *Resolution:* Adjusts the DPI (dots per inch) resolution of the configured plotter. Changing the DPI resolution changes the options available in the Dithering list. A higher resolution setting uses more memory and takes longer to plot than a lower resolution setting.
- *Dithering:* Specifies a dithering choice for non-pen-based plotters. Some dithering choices cause slower plotting.

**Raster Graphics (Non-Pen Plotters Only)** Specifies trade-offs between plotting speed and output quality when plotting raster objects. If you reduce the image quality, you increase output speed. If your system resources are limited, reducing image quality can reduce the chance of running out of memory

while plotting. These options are available only for raster devices, not pen plotters.

- *Raster and Shaded/Rendered Viewports:* Specifies a position on the slider that balances output quality with memory and plotting speed when plotting raster images and shaded/rendered viewports. Position the slider at None to disable raster image printing. Degrading the image quality lets you plot in less time. Position the slider at Best for the best output at the expense of memory and plotting speed.
- *OLE:* Specifies a position on the slider that balances output quality with memory use and plotting speed when plotting OLE objects. Position the slider at None to disable OLE object printing. Degrading the image quality lets you plot in less time. Position the slider at Best for the best output at the expense of memory and plotting speed.
- *Trade-off:* Specifies where to compromise quality if you can't output at the highest quality. Move the slider to diminish resolution and color.

**TrueType Text** On Windows system printers, specifies whether to plot TrueType text as a graphic image or as text. Plotting as a graphic guarantees that the text is printed as displayed, at the expense of slowing down the plotter and using more memory. Plotting as TrueType text prints faster and uses less memory; the plotter may use a different font for printing.

- *TrueType as Text:* Plots TrueType text as text.
- *TrueType as Graphics:* Plots TrueType text as graphics.

**Merge Control** On raster plotters, controls the appearance of lines that cross. Merge control is not effective if your plotter is configured to plot everything as black or if you are using PostScript language.

- *Lines Overwrite:* Uses the last plotted line to obscure the lines under it. Only the topmost line is visible at the intersection.
- *Lines Merge:* Merges the colors of crossing lines.

---

**NOTE** Merge control may appear as an option for system printers that do not actually support the feature. Please check your printer's documentation to determine if merge control is supported.

---

### Custom Properties Node

Modifies the device-specific properties for the plotter configuration. The settings for each plotter vary. If the plotter manufacturer has not included a

Custom Properties dialog box for the device driver, the Custom Properties option is disabled. For some drivers, such as ePlot, this is the only tree view option that is displayed. For Windows system printers, most of the device-specific settings are made in this dialog box. For more information about the custom properties settings for your device, choose Help in the Custom Properties dialog box.

### Initialization Strings Node (for Non-System Plotters Only)

Sets pre-initialization, post-initialization, and termination ASCII text plotter strings, which send commands to a plotting device before and after the program initializes the device and after plotting is complete.

If you are plotting to an unsupported plotter in emulation mode, you can specify ASCII text initialization strings that prepare the plotter for printing, set device-specific options, and restore the plotter to its original state. You can also use initialization strings to turn on or off a plotting device feature that is not supported by the program.

The text string is sent literally, except for a back slash (\). Use a back slash followed by a three-digit number (taken from the ASCII table), for example, \027, to specify binary (unprintable) characters such as the escape character. \027 is interpreted and sent as a single character whose value is 27. The number 27 is the escape character. For example, \27%-12345X@PJL ENTER LANGUAGE = PostScript\10 sends an HP PJL command to a dual-language laser printer before it's initialized and switches the printer into PostScript mode. The \027 sends an escape character and the \010 sends a line feed character. The remainder of the text string is sent literally. It is best to use three decimal digits for binary characters, so you'll need to add leading zeros as necessary.

Initialization strings should be used by advanced users only.

**Pre-Initialization** Forces a plotter to emulate another plotter by sending a pre-initialization ASCII text string to the plotter before it is initialized. Use a back slash followed by a three-digit number (taken from the ASCII table), for example, \027, to specify binary (unprintable) characters such as the escape character.

**Post-Initialization** Sets a device-specific option that is not supported elsewhere in the program. Specify a post-initialization ASCII text string that is sent to the plotter after it is initialized. Use a back slash followed by a three-digit number (taken from the ASCII table), for example, \027, to specify binary (unprintable) characters such as the escape character.

**Termination** Restores the printer to its original state after plotting. Specify a termination ASCII text string that is sent to the plotter after plotting is

complete. Use a back slash followed by a three-digit number (taken from the ASCII table), for example, \027, to specify binary (unprintable) characters such as the escape character.

### User-Defined Paper Sizes & Calibration Node

Attaches a PMP file to the PC3 file, calibrates the plotter, and adds, deletes, revises, or filters custom paper sizes. You can also modify standard paper sizes. This node accesses the Plotter Calibration and Custom Paper Size wizards. If the plotter you are using has been calibrated, a Plotter Model Parameter (PMP) file contains that calibration information. If the PMP file is not already attached to the PC3 file you are editing, you must create that association so you can use the PMP file. If the plotter was calibrated from within the Add-a-Plotter wizard while creating the current PC3 file, the PMP file is already attached. Use the PMP File Name option under User-defined Paper Sizes & Calibration to attach a PMP file to, or detach the PMP file from, the PC3 file you are editing.

**Custom Paper Sizes (Nonsystem Printers Only)** Creates a customized paper size or changes the printable area of a standard or nonstandard paper size. With the Custom Paper Size wizard you can create a new paper size, or select from a list of available paper sizes (from a PMP file) if the plotter is not a Windows system printer. If the plotter is a Windows system printer, use Custom Properties. You can add, delete, and edit your paper size.

Each plotter has a maximum printable area determined by where it grips the paper and how far the pen shuttle can reach. If you are creating a paper size that is larger than the paper sizes offered in the Plotter Configuration Editor, verify that the plotter is capable of plotting the new dimensions.

- *Add:* Starts the Custom Paper Size wizard. When you add a paper size, you can either create a new paper size from scratch or create a new one based on the listing of available paper sizes for the selected configured plotter. The new paper size is a user-defined size, not a standard size.
- *Delete:* Deletes the selected custom paper size from the list.
- *Edit:* Starts the Custom Paper Size wizard, where you can modify the selected paper size. You can change any of the custom paper size settings.

**NOTE** For Windows system printers, you can change the paper size settings and create custom paper sizes only in the Custom Properties dialog box.

**Modify Standard Paper Sizes** Adjusts the printable area for standard paper sizes to match the printer's capabilities. (You can't create custom paper sizes for Windows system printers using the Plotter Configuration Editor.)

- *List of Standard Paper Sizes:* Displays the available set of standard paper sizes.
- *Modify:* Starts the Custom Paper Size wizard. You can modify Printable Area and File Name. The new paper size is a user-defined size, not a standard size.

**Filter Paper Sizes** Filters the list of paper sizes displayed for the plotting device selected in the Plot and Page Setup dialog boxes. The list of paper sizes is displayed on the Plot Settings tab in the Plot dialog box and on the Layout Settings tab in the Page Setup dialog box. Select the paper sizes you want to display for this device.

- *Check All:* Hides all the paper sizes for the device.
- *Uncheck All:* Displays all the paper sizes for the device.

**Plotter Calibration** Starts the Plotter Calibration wizard. If you need to correct scaling discrepancies, you can adjust the plotter calibration using the Plotter Calibration wizard. See "Calibrate Plotters and Work with Custom Paper Sizes" in the *Driver and Peripheral Guide*.

---

**NOTE** You should perform a plotter calibration only if your drawings must be exactly to scale and your plotter or printer produces inaccurate plots. Plotter Calibration causes the program to rescale all plots sent to your plotter. If your plotter provides a calibration utility, it is recommended that you use it instead of the utility supplied with this program.

---

**PMP File** Attaches a PMP file to or detaches a PMP file from the PC3 file you are editing. Use the Detach button to break the association between the PMP file and the PC3 file.

- *Attach:* Attaches a PMP file to the PC3 file. You can reuse calibration and custom paper size data stored in the PMP file.
- *Save PMP:* Saves a PMP file to a new file in the *AutoCAD 2008\drv* folder.
- *Detach:* Detaches the PMP file associated with the PC3 file you are editing.

### Import

Imports file information from earlier versions of the program. If you have a PCP or PC2 file from an earlier version, you can import some of the

information in those files into a PC3 file. PC3 files store plotter name, port information, pen optimization level, paper size, and resolution.

#### Save As

Saves a PC3 File to a new file name.

#### Defaults

Restores the settings on the Device and Document Settings tab back to the default settings.

## Configure LPT Port Dialog Box

### Quick Reference

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

Specifies the time that the program waits for the plotter buffer to empty before sending more data. You specify how much time you want to elapse before you are prompted to cancel the plot. If your drawings are complex or your pen speed is very slow, set the timeout value higher than the default (30 seconds). If you begin to receive numerous timeout prompts, your timeout setting is probably too low.

#### Timeout

Specifies an amount of time to wait before retrying.

**Transmission Retry** Specifies the timeout value in milliseconds.

## Settings for COM Port Dialog Box

### Quick Reference

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

Specifies the baud rate, protocol, flow control, and hardware handshaking for serial ports on plotters that support these settings.

**NOTE** The settings on your plotter must match the settings in the program or you cannot plot.

**Baud Rate** Specifies the baud rate. Use the fastest available baud rate.

**Protocol** Displays the protocol settings available for your plotter. Use the protocol recommended by your device manufacturer. For more information see the documentation for your device.

**Flow Control** Sets the flow control for your plotter. The default flow control setting is XON/XOFF (software handshaking) for compatibility with previous versions of the program. If you select Hardware handshaking, you can specify additional settings in the Advanced Settings for COM Port dialog box (page 1093) that correspond to different pins on the RS232 connector.

# Advanced Settings for COM Port Dialog Box

## Quick Reference

File ➤ Plotter ManagerAt the Command prompt, enter plottermanager.
**plottermanager**

Specifies additional hardware handshaking settings for serial ports.

### CTS

Enables Clear To Send handshaking. This monitors pin 5 of a 25-pin serial port or pin 8 of a 9-pin serial port. CTS is an input bit, monitored for an output from the plotter or printer.

### DSR

Enables Data Set Ready handshaking. This monitors pin 6 of a 25-pin serial port, or pin 6 of a 9-pin serial port. DSR is an input bit, monitored for an output from the printer or plotter.

### RLSD

Enables Received Line Signal Detector handshaking. This is sometimes called DCD or Data Carrier Detect. It is an input pin that can be monitored for an

output from the plotter. It is on pin 8 of a 25-pin serial port and on pin 1 of a 9-pin serial port.

#### RTS

Controls the Request To Send output bit. This sends a signal to the printer or plotter on pin 4 of a 25-pin serial port or on pin 7 of a 9-pin serial port.

**Disabled** Disables the RTS line when the device is opened.

**Enabled** Enables the RTS line when the device is opened.

**Handshake** Enables RTS handshaking. The driver raises the RTS line when the input buffer is less than one-half full and lowers the RTS line when the buffer is more than three-quarters full.

**Toggle** Specifies that the RTS line is high if bytes are available for transmission. After all buffered bytes have been sent, the RTS line is low.

#### DTR

Controls the Data Terminal Ready output pin. This sends a signal to the printer or plotter on pin 20 of a 25-pin serial port or on pin 4 of a 9-pin serial port.

**Disabled** Disables the DTR line when the device is opened.

**Enabled** Enables the DTR line when the device is opened.

**Handshake** Enables DTR handshaking.

# PNGOUT

## Quick Reference

Saves selected objects to a file in a Portable Network Graphics format

**pngout**

The Create Raster File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the file name in the dialog box.

Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A Portable Network Graphics file is created that contains the objects you select. The file reflects what is displayed on the screen. Light glyphs that are displayed

in the drawing appear in the new file, even if the Plot Glyph property of the lights is set to No.

**NOTE** When the *FILEDIA* system variable is set to 0 (Off), prompts are displayed at the command prompt.

# POINT

## Quick Reference

Creates a point object

Draw 

Draw ➤ PointDoes not exist in the menus.

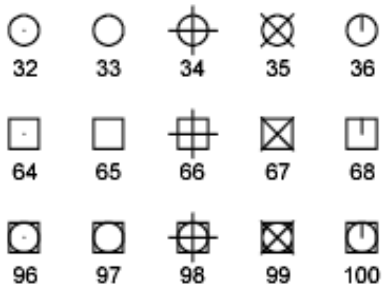
**point**

Specify a point:

Points can act as nodes to which you can snap objects. You can specify a full three-dimensional location for a point. The current elevation is assumed if you omit the *Z* coordinate value.

The *PDMODE* and *PDSIZE* system variables control the appearance of point objects. PDMODE values 0, 2, 3, and 4 specify a figure to draw through the point. A value of 1 specifies that nothing is displayed.


Specifying the value 32, 64, or 96 selects a shape to draw around the point, in addition to the figure drawn through it:




PDSIZE controls the size of the point figures, except for PDMODE values 0 and 1. A setting of 0 generates the point at 5 percent of the drawing area height. A positive PDSIZE value specifies an absolute size for the point figures. A negative value is interpreted as a percentage of the viewport size. The size of all points is recalculated when the drawing is regenerated.

After you change PDMODE and PDSIZE, the appearance of existing points changes the next time the drawing is regenerated.

# POINTLIGHT

## Quick Reference

Creates a point light

Lights 

View ➤ Render ➤ Light ➤ New Point LightAt the Command prompt, enter pointlight.

**pointlight**

Light panel (click to extend), Create a Point Light

Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*

If the LIGHTINGUNITS system variable is set to 0, the following prompt is displayed:

Enter an option to change [Name (page 1097)/Intensity (page 1097)/Status (page 1097)/shadoW (page 1098)/Attenuation (page 1099)/Color (page 1100)/eXit (page 1100)] <eXit>:

If the LIGHTINGUNITS system variable is set to 1 or 2, the following prompt is displayed:

Enter an option to change [Name (page 1097)/Intensity factor (page 1097)/Status (page 1097)/Photometry (page 1097)/shadoW (page 1098)/Attenuation (page 1099)/filterColor (page 1100)/eXit (page 1100)] <eXit>:

**NOTE** When the LIGHTINGUNITS system variable is set to 1 or 2, the Attenuation option has no affect on the creation of the light. It is only maintained for scripting compatibility.

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity/Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00-max float) <1.0000>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect

Enter status [oN/oFf] <On>:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived power per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:

**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i**, you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*>:

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command option.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFtmapped/softsAmpled] <Sharp>:
**Off** Turns off display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory to use to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness to use to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Linear, Disk, Rect, Sphere,Cylinder] <Sphere>:

Specify the sample size by entering **a.**

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v**.

Enter Shape Visibility [Yes/No]<No>:

### Attenuation

Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit]<eXit>:
**Attenuation Type** Controls how light diminishes over distance. The farther away an object is from a point light, the darker the object appears. Attenuation is also known as decay.

Enter attenuation type [None/Inverse linear/inverse Squared] <Inverse linear>:

- None. Sets no attenuation. Objects far from the point light are as bright as objects close to the light.
- Inverse Linear. Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the point light; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.
- Inverse Squared. Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the point light; at a distance of 4 units, light is one sixteenth as strong.

**Use Limits** Specifies whether to use limits or not.

Limits [oN/oFf] <Off>:

**Attenuation Start Limit** Specifies the point where light starts as an offset from the center of the light. The default is 0.

Specify start limit offset <1.0000>:

---

**NOTE** Attenuation start limits and end limits are not supported by the OpenGL driver (*wopengl9.hdi*), but the Direct 3D driver (*direct3d9.hdi*) does support end limits for attenuation but does not support start limits. To identify your driver, enter **3dconfig**, and click Manual Tune. Look at the selected Driver Name in the Manual Performance Tuning dialog box.

---

**Attenuation End Limit** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point. Setting an end limit increases performance where the effect of lighting is so minimal that the calculations are wasted processing time.

Specify end limit offset <10.0000>:

### Color/Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]<255,255,255>:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L) <0,0,100>:

**Color Book** Specifies a color from a color book.

Enter Color Book name:

### Exit

Exits the command.

# POLYGON

## Quick Reference

Creates an equilateral closed polyline

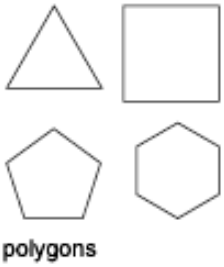
Draw 

Draw ➤ PolygonAt the Command prompt, enter polygon.

**polygon**

Enter number of sides <*current*>: *Enter a value between 3 and 1024 or press* ENTER

Specify center of polygon (page 1101) or [Edge (page 1102)]: *Specify a point (1) or enter* **e**



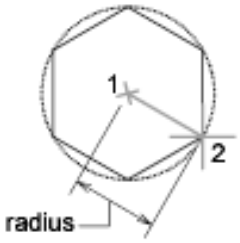

### Center of Polygon

Defines the center of the polygon.

Enter an option [Inscribed in circle/Circumscribed about circle] <*current*>: *Enter* **i** *or* **c** *or press* ENTER

**Inscribed in Circle** Specifies the radius of a circle on which all vertices of the polygon lie.

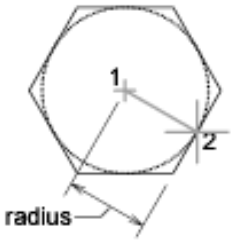
Specify radius of circle: *Specify a point (2) or enter a value*




Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

**Circumscribed about Circle** Specifies the distance from the center of the polygon to the midpoints of the edges of the polygon.

Specify radius of circle: *Specify a distance*

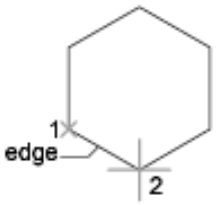



Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

### Edge

Defines a polygon by specifying the endpoints of the first edge.

Specify first endpoint of edge: *Specify a point (1)*
Specify second endpoint of edge: *Specify a point (2)*



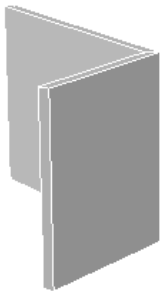
# POLYSOLID

## Quick Reference

Creates a 3D polysolid

Modeling
Draw ➤ Modeling ➤ PolysolidAt the Command prompt, enter Polysolid.
**polysolid**
3D Make panel, Polysolid

With the POLYSOLID command, you can convert an existing line, 2D polyline, arc, or circle to a solid with a rectangular profile. A polysolid can have curved segments, but the profile is always rectangular by default.



You can draw a solid with POLYSOLID just as you would a polyline. The *PSOLWIDTH* system variable sets the default width for the solid. The *PSOLHEIGHT* system variable sets the default height for the solid.

Specify start point or [Object (page 1103)/Height (page 1104)/Width (page 1104)/Justify (page 1104)] <Object>: *Specify a start point for the profile of the solid, press* ENTER *to specify an object to convert to a solid, or enter an option*
Specify the next point (page 1104) or [Arc (page 1105)/Undo (page 1106)]: *Specify the next point for the profile of the solid, or enter an option*

### Object

Specifies an object to convert to a solid. You can convert:

- Line

- Arc
- 2D polyline
- Circle

Select object: *Select an object to convert to a solid*

### Height

Specifies the height of the solid. The default height is set to the current *PSOLHEIGHT* setting.

Specify height <default>: *Specify a value for the height, or press* ENTER *to specify the default value*

The specified height value will update the *PSOLHEIGHT* setting.

### Width

Specifies the width of the solid. The default width is set to the current *PSOLWIDTH* setting.

Specify width <current>: *Specify a value for the width by entering a value or specifying two points, or press* ENTER *to specify the current width value*

The specified width value will update the *PSOLWIDTH* setting.

### Justify

Sets the width and height as the solid to be left, right, or center justified when defining the profile with the command. The justification is based on the starting direction of the first segment of the profile.

Enter justification [Left/Center/Right] <Center>: *Enter an option for the justification or press* ENTER *to specify center justification*

### Next Point

Specify the next point or [Arc/Close/Undo]: *Specify the next point for the profile of the solid, enter an option, or press* ENTER *to end the command*

### Arc

Adds an arc segment to the solid. The default starting direction of the arc is tangent to the last drawn segment. You can specify a different starting direction with the Direction option.

Specify endpoint of arc or [Close/Direction/Line/Second point/Undo]: *Specify an endpoint or enter an option*
**Close** Closes the solid by creating a line or arc segment from the last point specified to the starting point of the solid. At least two points must be specified to use this option.

**Direction** Specifies a starting direction for the arc segment.

Specify the tangent direction from the start point of arc: *Specify a point*

Specify endpoint of arc: *Specify a point*

**Line** Exits the Arc option and returns to the initial POLYSOLID command prompts.

**Second Point** Specifies the second point and endpoint of a three-point arc segment.

Specify second point on arc: *Specify a point*

Specify end point of arc: *Specify a point*

**Undo** Removes the most recent arc segment added to the solid.

### Close

Closes the solid by creating a line or arc segment from the last point specified to the starting point of the solid. At least three points must be specified to use this option.

### Undo

Removes the most recent arc segment added to the solid.

### Arc

Adds an arc segment to the solid. The default starting direction of the arc is tangent to the last drawn segment. You can specify a different starting direction with the Direction option.

Specify endpoint of arc or [Close/Direction/Line/Second point/Undo]: *Specify an endpoint or enter an option*

#### Close

Closes the solid by creating a linear or arc segment from the last vertex to the start of the solid.

#### Direction

Specifies a starting direction for the arc segment.

Specify the tangent direction from the start point of arc: *Specify a point*
Specify endpoint of arc: *Specify a point*

#### Line

Exits the Arc option and returns to the initial POLYSOLID command prompts.

#### Second Point

Specifies the second point and endpoint of a three-point arc segment.

Specify second point on arc: *Specify a point*
Specify end point of arc: *Specify a point*

#### Undo

Removes the most recent arc segment added to the solid.

#### Undo

Removes the most recent segment added to the solid.

# PRESSPULL

## Quick Reference

Presses or pulls bounded areas

Modeling 
**presspull**
3D Make panel, Press-pull
Click inside bounded areas to press or pull.

You can press or pull any of the following types of bounded areas:

- Any area that can be hatched by picking a point (with zero gap tolerance)
- Areas enclosed by crossing coplanar, linear geometry, including edges and geometry in blocks
- Closed polylines, regions, 3D faces, and 2D solids that consist of coplanar vertices
- Areas created by geometry (including edges on faces) drawn coplanar to any face of a 3D solid

# PREVIEW

## Quick Reference

Shows how the drawing will look when it is plotted

Standard 
File ➤ Plot PreviewAt the Command prompt, enter preview.
**preview**

PREVIEW displays a full-page preview of the current drawing. The preview is based on the current plot configuration, as defined by the settings in the Page Setup (page 1005) or Plot (page 1053) dialog box.

The cursor changes to a magnifying glass with plus (+) and minus (-) signs. Dragging the cursor toward the top of the screen while holding down the pick button enlarges the preview image. Dragging toward the bottom of the screen reduces the preview image.

The Preview window toolbar (page 1108) provides additional options.

**NOTE** When Texture Compression is turned on, there is a reduction in the quality of the images in the drawing when they are previewed. Texture Compression does not affect viewports that are rendered. To identify if Texture Compression is enabled, enter **3dconfig**, and click Manual Tune. Look at the Hardware Effects List in the Manual Performance Tuning dialog box.

**NOTE** When hardware acceleration is disabled or is enabled, but does not support Shadows, it is possible to preview a drawing that contains shaded viewports with unsupported hardware effects by the graphics card through software emulation. To enable the software emulation of hardware effects that are not supported by your graphics card, enter **3dconfig**, and click Manual Tune. In the Manual Performance Tuning dialog box, click Emulate unsupported hardware effects in software when plotting. The unsupported effects will not appear in the viewport in real-time, but will appear when previewing the drawing.

### Preview Window Toolbar

Provides options in the Preview window.

**Plot** Plots the drawing as it is displayed in the full-sheet preview, and then exits Plot Preview.

**Pan** Displays the pan cursor, a hand cursor that you can use to pan the preview image. Hold down the pick button and drag the cursor in any direction. The pan cursor stays active until you click another button.

**Zoom** Displays the zoom cursor, a magnifying-glass cursor that you can use to enlarge or reduce the preview image. To enlarge the image, hold down the pick button and drag the cursor toward the top of the screen. To reduce the image, hold down the pick button and drag the cursor toward the bottom of the screen.

**Zoom Window** Zooms to display a specified window. Zoom Window works with both the zoom cursor and the pan cursor.

**Zoom Original** Restores the initial full-sheet preview. Zoom Original works with both the zoom cursor and the pan cursor.

**Close Preview Window** Closes the Preview window.

# PROPERTIES

## Quick Reference

Controls properties of existing objects

Standard 
Modify ➤ Properties.Does not exist in the menus.
Select the objects whose properties you want to view or modify, right-click in the drawing area, and click Properties.
Pointing Device ➤ Double-click most objects
**properties**

The Properties palette (page 1109) is displayed. The Properties palette lists properties of the selected object or set of objects.

You can also view or modify properties of third-party application objects that are based on AutoCAD application programming interface (API) standards.

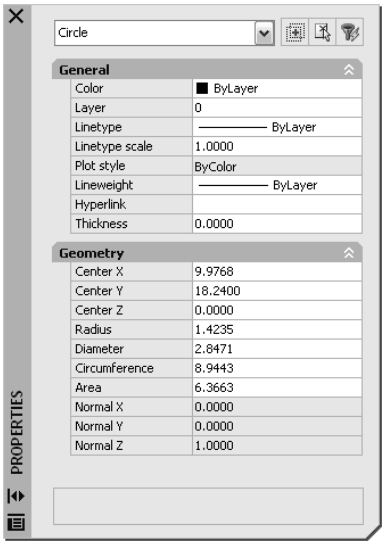
There are eight general properties (page 1112) common to all objects. All other object properties are specific to the type of object.

# Properties Palette

## Quick Reference

Standard
Modify ➤ PropertiesDoes not exist in the menus.

Select the objects whose properties you want to view or modify, right-click in the drawing area, and click Properties.
Pointing device ➤ Double-click most objects
**properties**




Displays the properties of the selected object or set of objects.

- When more than one object is selected, the Properties palette displays only those properties common to all objects in the selection set.
- When no objects are selected, the Properties palette displays only the general properties of the current layer and layout, the name of the plot style table attached to the layer, the view properties, and information about the UCS.

You can specify a new value to modify any property that can be changed. Click the value and use one of the following methods:

- Enter a new value.
- Click the down arrow at the right and select a value from the list.
- Click the Pick Point button to use the pointing device to change a coordinate value.
- Click the QuickCalc calculator button to calculate a new value.
- Click the left or right arrow to increase or decrease the value.

- Click the {...} button and change the property value in a dialog box.

**Object Type** Displays the type of object that is selected.

**Toggle Value of PICKADD Sysvar** Turns the PICKADD system variable on (1) and off (0). When PICKADD is on, each object selected, either individually or by windowing, is added to the current selection set. When PICKADD is off, selected objects replace the current selection set.

**Select Objects** Selects desired objects using any selection method. The properties common to the selected objects are displayed in the Properties palette. You can then modify the properties of the selected objects in the Properties palette, or you can make other changes to the selected objects by entering an editing command.

**Quick Select** Displays the Quick Select dialog box (page 1152). Use Quick Select to create selection sets based on filtering criteria.

The following shortcut menu options are available when you right-click the title bar.

**Move** Displays a four-headed arrow cursor that you can use to move the palette. The palette does not dock.

**Size** Displays a four-headed arrow cursor that you can use to drag an edge or a corner to make the palette smaller or larger.

**Close** Closes the Properties palette.

**Allow Docking** Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. A docked window adheres to the side of the application window and causes the drawing area to be resized. Selecting this option also makes Anchor Right and Anchor Left available.

**Anchor Right/ Anchor Left** Attaches the Properties palette to an anchor tab base at the right or left side of the drawing area. The palette rolls open and closed as the cursor moves across it. When an anchored palette is open, its content overlaps the drawing area. An anchored palette cannot be set to stay open.

**Auto-hide** Causes a floating palette to roll open and closed as the cursor moves across it. When this option is cleared, the palette stays open.

**Transparency** Displays the Transparency dialog box (page 994).

**Description** Controls display of the description area at the bottom of the Properties palette. When this option is checked, descriptions of options in the

Properties palette are displayed. When this option is cleared, no descriptions are displayed.

# General Properties of Objects

## Quick Reference

The following general properties are common to most objects and custom objects (third-party application objects based on AutoCAD API standards).

**Color** Specifies the color for objects. Selecting Select Color in the color list displays the Select Color dialog box (see *COLOR*).

You can use the Select Color dialog box to define the color of objects by selecting from the 255 AutoCAD Color Index (ACI) colors, true colors, and color book colors.

**Layer** Specifies the current layer of the object. The list shows all layers in the current drawing (see *LAYER*).

**Linetype** Specifies the current linetype of the object. The list shows all linetypes in the current drawing (see *LINETYPE*).


**Linetype Scale** Specifies the linetype scale factor of the object (see *LTSCALE*).

**Plot Style** Lists NORMAL, BYLAYER, BYBLOCK, plus any plot styles contained in the current plot style table (see *PLOTSTYLE*).

**Lineweight** Specifies the lineweight of the object. The list shows all available lineweights in the current drawing (see *LWEIGHT*).

**Hyperlink** Attaches a hyperlink to a graphical object. If a description has been assigned to the hyperlink, the description is displayed. If no description has been assigned, the URL is displayed (see *HYPERLINK*).

**Thickness** Sets the current 3D thickness. This property does not apply to all objects (see *CHPROP*).

# Cell Border Properties Dialog Box

## Quick Reference

Pointing device ➤ With a table cell selected and the Properties palette open, click in the value cell for Border Lineweight or Border Color, and click the [...] button.

Sets the properties of the borders of table cells.

### Border Properties

Controls the properties of the borders of the selected table cells.

**Lineweight** Sets the lineweight to be used for borders that are displayed. If you use a heavy lineweight, you may have to change the cell margins.

**Linetype** Sets the linetype to be used for borders that are displayed. You can choose Other to display the Choose Linetype Dialog Box.

**Color** Sets the color to be used for borders that are displayed. You can choose Select Color to display the Select Color dialog box (page 251).

**Double Line** When checked, a double line border will be added to the selected cells.

**Spacing** Determines the spacing of double-line borders. The default value is .1800.

### Apply To

Specifies which borders of the selected table cell have the Border Properties settings applied.

**All Borders** Applies the border properties settings to all borders of the selected table cells.

**Outside Borders** Applies the border properties settings to the outside borders of the selected table cells.

**Inside Borders** Applies the border properties settings to the inside borders of the selected table cells.

**No Borders** Applies the border properties settings to none of the borders of the selected table cells.

#### Preview

Displays the results of the current settings. Click a gridline in the preview to apply properties to it.

**NOTE** Linetypes will not display in the preview window.

## Add Distance or Angle Value Dialog Box

### Quick Reference

Adds distance or angle values to a parameter's value set in a dynamic block definition.

**Distances or Angle to Add** Specifies values (separated by commas) to add to the value set.

**Distances or Angles List (Unlabeled)** Lists the distances or angles defined for the value set.

**Add** Adds the values specified in the Distances or Angle to Add box.

**Delete** Deletes the selected value.

## Lighting Properties

### Quick Reference

Sets the properties of the lights. Different properties area available depending on the lighting units (standard or photometric) and lighting type (Spotlight (page 1355), Pointlight (page 1096), or Weblight (page 1550)). Other lighting types such as Freespot (page 594), Targetpoint (page 1419), and Freeweb (page 599) display similar properties. By right-clicking on a light and clicking Properties, the Lighting Properties palette is displayed.

#### General Properties

Under the General panel, the following property settings are available:

**Name** Specifies the name of the light.

**Type (Light Distribution)** Specifies the type of light. Determines the distribution of light from the lamp. The type of lighting can be changed after the light has been added to the drawings.

- Spotlight - Default value for Spotlight and Freespot lights.
- Point - Default for Pointlight and Targetpoint lights.
- Web - Default for Weblight and Freeweb lights.

**On/Off Status** Indicates whether the light is on or off.

**Shadows** Indicates if the light is casting a shadow.

**Hotspot Angle (Spotlight and Freespot only)** Specifies the angle of the brightest cone of light. Settings can be calculated by using the QuickCalc Calculator (page 1155).

**Falloff angle (Spotlight and Freespot only)** Specifies the outer extremity of the light, where it meets the darkness.

**Intensity Factor** Magnifies the effect of the skylight.

**Filter Color** Specifies the secondary color of the light. Represents the color of a physical filter over the lamp. Default color is white.

When lighting is set to photometric units this represents a secondary color filter on the light. When lighting is set to generic lighting this represents the total color of the light.

**Plot Glyph** Allows the ability to plot the drawing with the glyphs on.

### Photometric Properties

Under the Photometric Properties panel, the following property settings are available:

**Lamp Intensity** Specifies the inherent brightness of the light. Specifies the intensity, flux, or illuminance of the lamp. Default units are candela. Activates the Lamp Intensity dialog box, (page 1119) where units can be modified.

**Resulting Intensity** Reports the final brightness of the light. This is determined by the product of the Lamp Intensity and the Intensity factor. This value is calculated in the Lamp Intensity Dialog box (page 1119). (Read-only)

**Lamp Color** Specifies the inherent color of the light in Kelvin temperature or standard. The button activates the Lamp Color dialog box (page 1120).

**Resulting Color** Reports the final color of the light. This is determined by a combination of the Lamp Color and the Filter Color. (Read-only)

### Photometric Web

Under the Photometric Web panel, the following property settings are available under the Weblight and Freeweb types of lights:

**Web File** Specifies the data file describing the intensity distribution of the light.

**Web Preview** Displays a 2D slice through goniometric data.

### Web Offsets

Under the Web offsets panel, the following property settings are available under the Weblight and Freeweb types of lights:

**Rotation X** Specifies a rotational offset of the web about the optical X axis. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Rotation Y** Specifies a rotational offset of the web about the optical Y axis. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Rotation Z** Specifies a rotational offset of the web about the optical Z axis. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

### Geometry

Under the Geometry panel, the following property settings are available:

**Position X** Specifies the X coordinate position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Position Y** Specifies the Y coordinate position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Position Z** Specifies the Z coordinate position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Target X (Spotlight, Targetpoint, and Weblight only)** Specifies the X coordinate target position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Target Y (Spotlight, Targetpoint, and Weblight only)** Specifies the Y coordinate target position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Target Z (Spotlight, Targetpoint, and Weblight only)** Specifies the Z coordinate target position of the light. Settings can be calculated by using the QuickCalc Calculator (page 1155) or by selecting a point.

**Targeted** Specifies if the light displays a target grip for orienting the light. No is the default for Freespot, Pointlight, and Freeweb. Yes is the default for Spotlight, Targetpoint, and Weblight.

### Attenuation

In the real world, the intensity of light diminishes over distance. Objects far from the light source appear darker than objects near the source. This effect is known as attenuation. Attenuation is available under standard lighting workflow only. Under the Attenuation panel the following property settings are available:

**Type** Controls how light diminishes over distance. The farther away an object is from a spotlight, the darker the object appears. Attenuation is also known as decay.

- Inverse Linear (Standard lights only). Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the point light; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.
- Inverse Square (Photometric lights). Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the spotlight; at a distance of 4 units, light is one sixteenth as strong.
- None (Standard lights only). Sets no attenuation. Objects far from the point light are as bright as objects close to the light.

**Use Limits (Standard lights only)** Specifies whether to use limits. The default is No.

**Start Limit Offset (Standard lights only)** Specifies the point where light starts as an offset from the center of the light. The default is 1.

**End Limit Offset (Standard lights only)** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point.

### Rendered Shadow Details

Under the Rendered Shadow Details panel, the following property settings are available:

**Type** Specifies the type of shadow cast by the light.

- Soft (shadow map). Sets the type to Soft. This selection activates additional options for Map size and Softness.
- Sharp (default). Sets the rendered shadow to sharp.
- Soft (sampled). Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the spotlight; at a distance of 4 units, light is one sixteenth as strong.

**Map Size (Soft shadow map type only)** Specifies the size of the shadow map.

**Softness (Soft shadow map type only)** Specifies the softness or fuzziness of the shadow-mapped shadow.

**Samples (Soft sampled type only)** Specifies the number of shadow rays for the light. Settings can be calculated by using the QuickCalc Calculator (page 1155).

**Visible Render (Soft sampled type only)** Specifies whether the light shape is actually rendered. The default is No.

**Shape (Soft sampled type only)** Specifies the shape of the lamp bulb. For the Spotlight distribution type selection under the General panel, options are Rectangle (default) and Disk. For Point and Web types the options are Linear, Rectangle, Disk, Cylinder and Sphere (default).

**Length (Soft sampled type only)** Specifies spacial dimension of shadow shape for the length of the shadow. Settings can be calculated by using the QuickCalc Calculator (page 1155).

**Width (Soft sampled type only)** Specifies spacial dimension of shape for the width of the shadow. Settings can be calculated by using the QuickCalc Calculator (page 1155).

**Radius (Soft sampled type only)** Specifies spacial radius dimension of the shape selection of disk, cylinder, or sphere. Settings can be calculated by using the QuickCalc Calculator (page 1155).

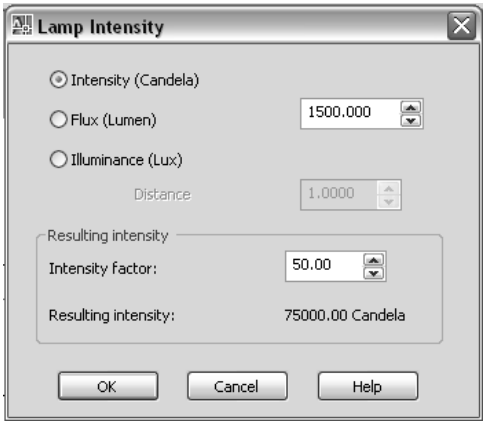
# Lamp Intensity Dialog Box

## Quick Reference

Modifies the brightness of a lamp.

### Lamp Intensity Property Settings

Dialog box for setting the Lamp Intensity for a photometric light.




**Lamp Intensity** Represents the brightness of a lamp. More specifically, it represents the luminous intensity, or power in a particular direction. It is specified in candela by default (in both the SI and American units systems). Must select from one of the following radial buttons:

- **Intensity (Candela)**. Specifies the number of candelas (cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction).
- **Flux (Lumen).** Represents the rate of total energy leaving the lamp. It is specified in lumens (SI and American). Mathematically, the flux is the integral of the luminous intensity over the sphere. The calculation of flux depends on the distribution of intensities. For a point light with constant intensity, the flux is simply the product of the intensity and the solid angle of a sphere: 4 Pi * Intensity. For a spot light, the flux is the product of the intensity and the solid angle of the hotspot cone, plus the incremental solid angle of the fall-off region. For a weblight, there isn't any analytical formula. The flux is obtained by numerically integrating the intensities provided in the web file.

- **Illuminance (Foot-candles).** Represents the energy per area arriving at a surface (Area-flux-density). It is specified in lux (SI) and foot-candles (American). For a near light, because the light rays are diverging, you have to talk about the illuminance at a specific distance from the lamp. So this requires an extra control to specify this distance and an additional affordance in the viewport to show the distance.

**Intensity Input Control** Controls the intensity value.

**Distance** When Illuminance is selected, displays the lux as a distance glyph in the viewport for the given light. Note: the text for the illuminance radio button will depend on the current lighting units. For the SI system, this will be "Illuminance (Lux)", for the American system it will be "Illuminance (Footcandles).The distance field represents a world distance and should be formatted in the current length unit corresponding to the current lighting units. The distance field represents a world distance and should be formatted in the current length unit corresponding to the current insertion units controlled by the INSUNITS systems variable.

**Resulting Intensity** Specifies the type of light. Determines the distribution of light from the lamp. The type of lighting can be changed after the light has been added to the drawings.

- **Intensity Factor.** Specifies the intensity factor.
- **Resulting Intensity.**Displays the resulting intensity of a light as the product of the Lamp Intensity value and the Intensity Factor. The value is read-only.

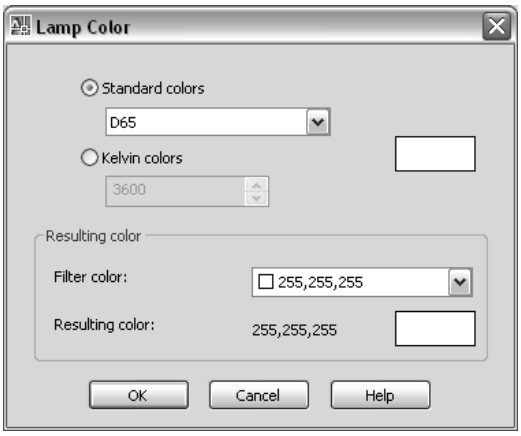
# Lamp Color Dialog Box

## Quick Reference

Provides settings for the lamp color.

### Lamp Color Settings

Dialog box for setting the Lamp Color for a photometric light.




**Color Type** In photometric mode specifies lamp color either as a CIE standard illuminant (D65 standard daylight) or as a Kelvin color temperature.

- **Standard Colors.** Provides a fixed list of standard colors (spectra).
- **Kelvin Colors.** Specifies a single temperature value for the color. (The color corresponds to the glow of an ideal blackbody at that temperature.) Kelvin temperature is a number in the range of 1000-20000.

**Resulting Color** Shows the result of modulating the lamp color by the Filter color. A text field displays the RGB component values of the resulting color.

- **Filter Color.** Provides a drop-down list for selecting a standard color or launching a standard color selection dialog box.
- **Resulting Color.** Shows the resulting color of a light that is the product of the lamp color and the filter color.

# PROPERTIESCLOSE

## Quick Reference

Closes the Properties palette

**propertiesclose**

The Properties palette closes.

# PSETUPIN

## Quick Reference

Imports a user-defined page setup into a new drawing layout

**psetupin**

The Select Page Setup From File dialog box (a standard file selection dialog box (page 931)) is displayed in which you can select the drawing *(.dwg)*, template (*.dwt*), or drawing interchange format (*.dxf*) file whose page setups you want to import.

If *FILEDIA* is set to 0 (zero) and you enter **-psetupin** at the command prompt, PSETUPIN displays command prompts (page 1122).

When you select the drawing file that you want to use, the Import Page Setups dialog box (page 1013) is displayed.

# -PSETUPIN

## Quick Reference

If you enter **-psetupin** at the command prompt, the following PSETUPIN command prompts are displayed.

Enter file name:
Enter user defined page setup to import or [?]:

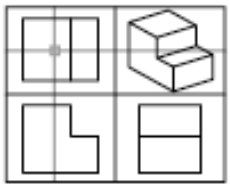
# PSPACE

## Quick Reference

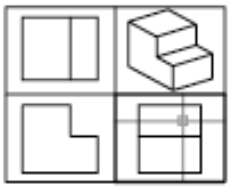
Switches from a model space viewport to paper space

**pspace**

The program switches from model space to paper space when you are working on a layout tab.



paper space



model space

On the layout tab, use paper space to create a finished layout of a drawing for printing. As part of designing your layout, you create layout viewports, which are windows containing different views of the model. By switching from paper space to model space (see *MSPACE*), you can edit the model and views within the current layout viewport.

You can make a viewport current by double-clicking inside it. You can switch to paper space by double-clicking an area of the paper space layout that is not within a viewport.

You can also switch between model space and paper space by choosing Model or Paper on the status bar.

# PUBLISH

## Quick Reference

Publishes drawings to DWF files or plotters

Standard 

File ➤ PublishAt the Command prompt, enter publish.

**publish**

The Publish dialog box (page 1124) is displayed. Click Publish to begin publishing the current drawing sheets to either a DWF file or a plotting device.

To display information about the published sheets, click the Plotting Details Report Available icon in the tray on the right side of the status bar. Clicking this icon opens the Plot and Publish Details dialog box (page 1516), which provides information about your completed plot and publish jobs. This information is also saved to the Plot and Publish log file. The shortcut menu for this icon also provides an option to view the most recently published Design Web Format™(DWF™).

If you enter **+publish** at the command prompt, you can select an existing Drawing Set Descriptions (DSD) file from the Select List of Sheets dialog box (a standard file selection dialog box (page 931)). The Publish dialog box opens and displays the drawing set saved in the DSD file under Sheets to Publish.

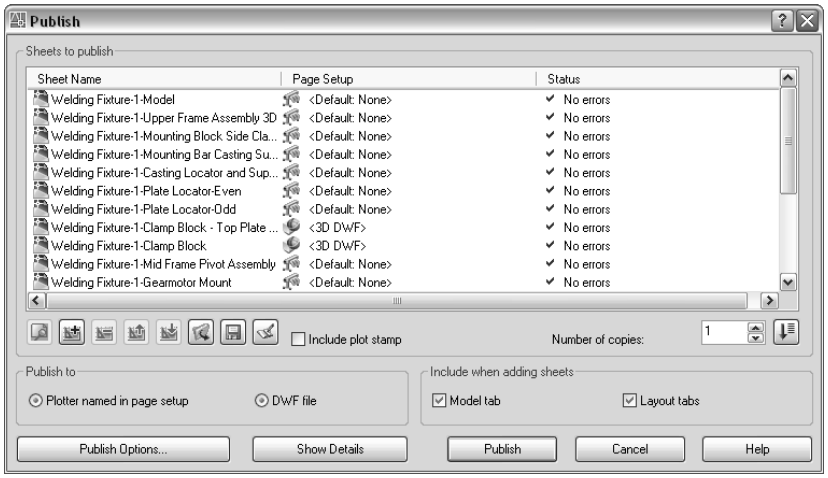
With the *FILEDIA* system variable set to 0, if you enter **-publish** at the command prompt, options are displayed at the command prompt (page 1134).

# Publish Dialog Box

## Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Specifies drawing sheets that you can assemble, reorder, rename, copy, and save for publishing as a multisheet drawing set. You can publish the drawing set to a DWF file or send it to the plotter named in the page setup for hardcopy output or as a plot file. You can save this list of drawing sheets as a DSD (Drawing Set Descriptions) file. Saved drawing sets can replace or be appended to the current list for republishing.



### Sheets to Publish

Contains the list of drawing sheets to be included for publishing. Click the page setup to change it. Use the shortcut menu to add sheets or make other changes to the list.

**NOTE** The drawing sheets that populate the Sheets to Publish list are controlled by the *PUBLISHALLSHEETS* system variable.

**Sheet Name** Combines the drawing name and the layout name with a dash (-). Includes the Model tab only if the Include Model When Adding Sheets option is selected. You can copy sheets by clicking Copy Selected Sheets on the shortcut menu. You can change the name shown in Sheet Name by clicking Rename Sheet on the shortcut menu. Drawing sheet names must be unique within a single DWF file. The shortcut menu also provides an option for removing all the sheets from the list.

**Page Setup / 3D DWF** Displays the named page setup for the sheet. You can change the page setup by clicking the page setup name and selecting another page setup from the list. Only Model tab page setups can be applied to Model tab sheets, and only paper space page setups can be applied to paper space layouts. Select Import to import page setups from another DWG file through the Import Page Setups for Publishing dialog box (a standard file selection dialog box).

You have the option to set the page setup for model space sheets to 3D DWF. The 3D DWF option is not available for layout entries in the sheet list.

**Status** Displays the status of the sheet when it is loaded to the list of sheets.

### Preview

Displays the drawing as it will appear when plotted on paper by executing the *PREVIEW* command. To exit the print preview and return to the Publish dialog box, press ESC, press ENTER, or right-click and then click Exit on the shortcut menu.

**NOTE** The Preview button is inactive when a sheet has its page setup set to 3D DWF.

### Add Sheets

Displays the Select Drawings dialog box (a standard file selection dialog box (page 931)), in which you can select drawings to add to the list of drawing

sheets. The layout names from those files are extracted, and one sheet is added to the list of drawing sheets for each layout and model.

The initial drawing sheet names are constructed from the base drawing name and the layout name or the word Model separated by a dash (-).

### Remove Sheets

Deletes the selected drawing sheets from the list of sheets.

### Move Sheet Up

Moves the selected drawing sheets up one position in the list.

### Move Sheet Down

Moves the selected drawing sheets down one position in the list.

### Load Sheet List

Displays the Load Sheet List dialog box (a standard file selection dialog box (page 931)), in which you can select a DSD file or a BP3 (Batch Plot) file to load. Displays the Replace or Append dialog box if a list of drawing sheets is present in the Publish Drawing Sheets dialog box. You can either replace the existing list of drawing sheets with the new sheets or append the new sheets to the current list.

### Save Sheet List

Displays the Save List As dialog box (a standard file selection dialog box (page 931)), in which you can save the current list of drawings as a DSD file. DSD files are used to describe lists of drawing files and selected lists of layouts within those drawing files.

### Plot Stamp Settings

Displays the Plot Stamp dialog box (page 1070), in which you can specify the information, such as drawing name and plot scale, that you want applied to the plot stamp.

### Include Plot Stamp

Places a plot stamp on a specified corner of each drawing and logs it to a file. The plot stamp data is specified in the Plot Stamp dialog box (page 1070).

While this check box remains active for sheets that are set for 3D DWF publishing, no plot stamp is added to 3D DWF files even if the check box is checked.

### Number of Copies

Specifies the number of copies to publish. If the Publish to DWF option is selected, the Number of Copies setting defaults to 1 and cannot be changed. If a sheet's page setup specifies to plot to file, then the number set in this option is ignored and a single plot file is created.

### Send the Sheets to the Plotter in Reverse Order

When selected, sends sheets to the plotter in reverse of default order. This option is available only if the Plotter Named in Page Setup option is selected.

### Publish To

Defines how to publish the list of sheets. You can publish to either a multisheet DWF file (an electronic drawing set) or to the plotter specified in the page setup (a paper drawing set or a set of plot files).

**Plotter Named in Page Setup** Indicates that the output devices given for each drawing sheet in the page setup will be used. When this switch is enabled, all non-3D DWF entries are published. Entries that are set to 3D DWF will be flagged by a warning in the Status column of the Sheets to Publish list.

**DWF file** Indicates that the list of sheets will be published as a DWF file.

### Include When Adding Sheets

Specifies whether the model and layouts contained in a drawing are added to the sheet list when you add sheets. At least one option must be selected.

**Model Tab** Specifies whether the model is included when drawing sheets are added.

**Layout Tabs** Specifies whether all layouts are included when drawing sheets are added.

### Publish Options

Opens the Publish Options dialog box (page 1128), in which you can specify options for publishing.

#### Show Details

Displays and hides the Selected Sheet Information and Selected Page Setup Information areas.

**Selected Sheet Information** Displays the following information about the selected sheet: source drawing, drawing location, and layout name.

**Selected Page Setup Information** Displays the following information about the selected page setup: plot device, plot size, plot scale, and details.

#### Publish

Begins the publishing operation. Creates one or more single-sheet DWF file or a single multi-sheet DWF file, or plots to a device or file, depending on the option selected in the Publish To area and the options selected in the Publish Options dialog box (page 1128).

To display information about the published drawing set, including any errors or warnings, click the Plotting Details Report Available icon in the status tray on the right side of the status bar. Clicking this icon displays the Plot and Publish Details dialog box (page 1516), which provides information about your completed plot and publish jobs. This information is also saved to the Plot and Publish log file. The shortcut menu for this icon also provides an option to view the most recently published DWF file.

## Publish Options Dialog Box

### Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Specifies options for publishing.

#### Current User or Current Sheet Set

Displays the name of the current user or the current sheet set. When the name of the current user is shown, changes made in the dialog box are saved in the current user's profile. When the name of the current sheet set is shown, changes made in the dialog box are saved with the sheet set.

### Default Output Directory (DWF and Plot-to-File)

Specifies the output folder location where Design Web Format (DWF) and plot files are saved when you publish drawing sheets.

**Location** Specifies where DWF and plot files are saved when you publish drawings.

### General DWF Options

Specifies options for creating a single-sheet DWF file.

**DWF Type** Specifies that a single-sheet DWF file or a single multisheet DWF file is generated for all the sheets listed in the Publish dialog box (page 1124).

**Password Protection** Specifies options for protecting DWF files with passwords.

**Disabled** Specifies that no password is required.

**Prompt for Password** Displays the DWF Password dialog box (page 1133) when you click Publish.

**Password** Specifies that published DWF files have a password applied to them. A recipient of a DWF file that has a password applied to it must have the password to open the DWF file.

**Specify Password** Specifies that the Password in the password box is applied to the DWF file. When you select this option, the Password box becomes available, and you must enter a password.

**NOTE** If you lose or forget the password, it cannot be recovered. Keep a list of passwords and their corresponding DWF file names in a safe place.

### Multi-sheet DWF Options

Specifies options for creating a multi-sheet DWF.

**Prompt for Name** Indicates that you will be prompted for a name and location for published multisheet DWF files.

**Specify Name** Prompts you for a name and location for published multisheet DWF files.

### DWF Data Options

Lists and allows you to specify the data that you can optionally include in the DWF file.

**Layer Information** Specifies whether layer information is included in the published DWF file.

**NOTE** Layer information for 3D DWF entries does not get published.

**Block Information** Specifies whether block property and attribute information is included in the published DWF files.

**NOTE** You must set block information to Include in order for block template file information to be available.

**Block Template File** Allows you to create a new block template (BLK) file, edit an existing block template file, or use the settings of a previously created block template file.

Create opens the Publish Block Template dialog box (page 1131), in which you can create a new block template.

Edit opens the Select Block Template dialog box (a standard file selection dialog box (page 931)), in which you can select an existing block template to modify.

**NOTE** When only 3D DWF files are queued for publishing, Block Information and Block Template File are set to N/A and cannot be changed.

### 3D DWF Options

Lists and allows you to specify the data that you can optionally include in 3D DWF publishing.

**Group By Xref Hierarchy** Arranges objects by xref hierarchy in the viewer. If set to No, xrefs are listed in the viewer like any other objects. Default = No.

**Publish With Materials** Most materials you have assigned to your model will be published to the 3D DWF file. If your material contains texture mapping, only the Diffuse Map is published along with its scale and orientation data. Default = Yes.

**NOTE** When only 2D DWF files are queued for publishing, all of the 3D DWF Options are set to N/A and cannot be changed.

# Publish Block Template Dialog Box (Publish)

## Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Specifies which blocks and their properties and attributes to include in the published DWF file.

### Current Template

Displays the name of the current block template.

### Block Source Drawings

Displays all block source drawings that have been loaded into the Publish Block Template dialog box.

### Add

Adds drawings to the list of block source drawings.

### Remove

Removes selected drawings from the list of block source drawings.

### Scan for Blocks

Scans block source drawings for unique definitions of blocks and their properties and attributes.

### Block Data to Publish

Displays block data to publish.

**Unique Blocks from Source Drawings** Displays unique blocks found in block source drawings.

**Check Blocks to Publish** Displays all scanned blocks found in block source drawings.

**Exclude Blocks Without Attributes** Excludes blocks without attributes defined.

**Properties of Selected Blocks** Displays properties of selected blocks found in block source drawings.

**Check Properties to Publish** Displays the union of properties of the current selection of blocks.

**Exclude General Block Properties** Excludes all general block properties from the list display.

#### Options

Opens the Block Template Options dialog box where you can specify additional settings.

#### Save

Saves the block template file.

## Block Template Options Dialog Box (Publish)

### Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Specifies optional settings to include blocks within blocks and to include blocks within xrefs in the published DWF file.

**Include Nested Blocks** Includes blocks within blocks in the published DWF file.

**Include Blocks in Xrefs** Includes blocks within xrefs in the published DWF file.

# Changes to a Printer Configuration File Dialog Box (Publish)

## Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Notifies you that you have made changes to an existing plotter configuration (PC3) file. You can cancel your changes, click OK to apply the changes for the current publish operation only, or save the changes to the DWF PC3 file.

**Apply Changes for the Current Publish Operation Only** Uses the changes you've made to the PC3 file in the current publish operation but does not save them to the PC3 file.

**Save Changes to the DWF PC3 File** Specifies that the changes you made in the Plotter Configuration Editor are saved to the DWF PC3 file.

# DWF Password Dialog Box

## Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Specifies the password to apply to the DWF file. DWF passwords are case sensitive. The password or phrase can be made up of letters, numbers, punctuation, or non-ASCII characters.

**WARNING** If you lose or forget the password, it cannot be recovered. Keep a list of passwords and their corresponding DWF file names in a safe place.

When you enter a password and click OK, the Confirm DWF Password dialog box (page 1134) is displayed.

## Confirm DWF Password Dialog Box

### Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Provides a space to confirm the password that you entered in the DWF Password dialog box (page 1133). If the two passwords do not match, you must click Publish again to reenter the correct password.

## Publish Job Progress Dialog Box

### Quick Reference

Standard
File ➤ PublishAt the Command prompt, enter publish.
**publish**

Displays information about the status and progress of your publish job if background publishing is turned off in the Options dialog box (page 946), Plot and Publish tab (page 961).

**Cancel Sheet** Cancels publishing of the sheet currently being processed.

**Cancel Job** Cancels the publish job.

## PUBLISH Command Line

### Quick Reference

**-publish**

The command line version of PUBLISH provides a scriptable interface for publishing drawing sheets. It also provides a simplified method for regenerating DWF files or plotted output from existing DSD files.

With the FILEDIA system variable set to 0 (or if you are running a script or other automation), enter **-publish** or **+publish** at the Command prompt. The following prompt is displayed:

Enter name of sheet list <path file name.dsd>: *Press* ENTER *for the current drawing or enter the sheet list name*

**NOTE** The bracketed text <DSD file name> contains the name of the DSD file used when this command was last run. The bracketed text area is blank if no previous name exists. If you enter a tilde (~) for the file name, the standard file selection dialog box is displayed regardless of the FILEDEA setting.

The file name is validated and the list of sheets is loaded.

When -PLOT, *PLOT*, -PUBLISH, and *PUBLISH* are used in a script (SCR file), the BACKGROUNDPLOT system variable value is ignored, and -PLOT, PLOT, -PUBLISH, and PUBLISH are processed in the foreground.

Once the publish job starts, each drawing sheet name is listed along with an appropriate message. Drawing sheets not found or not initialized are indicated with an error message.

When the publish job is complete, the details can be viewed in the Plot and Publish Details dialog box (page 1516).

A log file is written during the publish operation. The log file contains the drawing sheet name, layout name, full drawing name, path name, and status. The log file name is derived from the name of the sheet list file with CSV replacing the DSD file extension.

**NOTE** Any existing log file is overwritten by a new log file without any warning message being displayed.

# PUBLISHTOWEB

## Quick Reference

Creates HTML pages that include images of selected drawings

File ➤ Publish to Web
**publishtoweb**

The Publish to Web wizard (page 1136) is displayed.

## Publish to Web Wizard

### Quick Reference

File ➤ Publish to Web
**publishtoweb**

Creates a formatted HTML page. You can choose from a number of different formatting options that control the layout of your completed HTML page. After creating an HTML page, you can use the wizard to publish the page to an Internet or intranet location.

You can also use the wizard to modify existing HTML pages that were created using the PUBLISHTOWEB command.

You can customize the template you use for your web page. For information about customizing templates, see "Customize a Publish to Web Template" in the *Customization Guide*.

# PURGE

### Quick Reference

Removes unused named items, such as block definitions and layers, from the drawing

File ➤ Drawing Utilities ➤ PurgeAt the Command prompt, enter purge.
**purge**

The Purge dialog box (page 1136) is displayed.

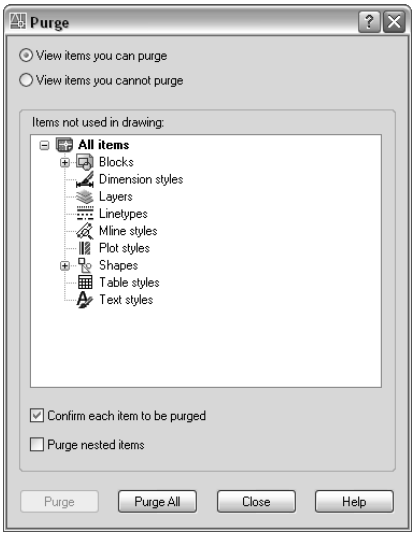
If you enter **-purge** at the command prompt, options are displayed at the command prompt (page 1138).

## Purge Dialog Box

### Quick Reference

File ➤ Drawing Utilities ➤ PurgeAt the Command prompt, enter purge.
**purge**

Displays items that can be purged.




### View Items You Can Purge

Switches the tree view to display a summary of named objects in the current drawing that you can purge.

**Items Not Used in Drawing** Lists the named objects that are not used in the current drawing and that can be purged. You can list the items for any object type by clicking the plus sign or by double-clicking the object type. You purge items by selecting the item to purge.

Purge Nested Items removes items only when you select one of the following options:

- All Items or Blocks in the tree view
- The Purge All button

**Confirm Each Item to Be Purged** Displays the Confirm Purge dialog box when you purge an item.

**Purge Nested Items** Removes all unused named objects from the drawing even if they are contained within or referenced by other unused named objects. The Confirm Purge dialog box is displayed, and you can cancel or confirm the items to be purged.

#### View Items You Cannot Purge

Switches the tree view to display a summary of named objects in the current drawing that you can't purge.

**Items Currently Used in Drawing** Lists named objects that cannot be removed from the drawing. Most of these objects are currently used in the drawing or are default items that cannot be removed. When you select individual named objects, information about why you can't purge the item is displayed below the tree view.

#### Tip

Displays information detailing why you can't purge the selected item.

#### Purge

Purges the selected items.

#### Purge All

Purges all unused items.

## -PURGE

### Quick Reference

If you enter **-purge** at the command prompt, the following PURGE command prompts are displayed.

Enter type of unused objects to purge
[Blocks/Dimstyles/LAyers/LTypes/MAterials/Plotstyles/SHapes/textSTyles/Mlinestyles/Tablestyles/VisualStyles/Regapps/All]:
*Enter an object type, enter* **r** *to purge unused applications, or enter* **a** *to purge all named object types but not applications*
Enter name(s) to purge <*>: *Enter one or more names, or press* ENTER *to purge all items*
Verify each name to be purged? [Yes / No]: *Enter* **y** *to verify each object that is to be purged, or enter* **n** *to purge the objects without any verification*

-PURGE removes only one level of reference. Repeat -PURGE until there are no unreferenced named objects. You can use PURGE or -PURGE at any time during a drawing session.

# PYRAMID

## Quick Reference

Creates a 3D solid pyramid

Modeling
Draw ➤ Modeling ➤ PyramidAt the Command prompt, enter pyramid.
**pyramid**
3D Make panel, Pyramid
4 sides *(default)*
Circumscribed *(default)*
Specify center point of base or [Edge (page 1139)/Sides (page 1140)]: *Specify a point or enter an option*

One of the following prompts is displayed:

Specify base radius or [Inscribed (page 1140)] <default>: *Specify a base radius, enter* **i** *to change the pyramid to circumscribed, or press* ENTER *to specify the default base radius value*
Specify base radius or [Circumscribed (page 1140)] <default>: *Specify a base radius, enter* **c** *to change the pyramid to circumscribed, or press* ENTER *to specify the default base radius value*

Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

After specifying the base radius and whether the pyramid is inscribed or circumscribed, the following prompt is displayed:

Specify height or [2Point (page 248)/Axis endpoint (page 1140)/Top radius (page 1140)] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Use the Top Radius option to create a pyramid frustum.

### Edge

Specifies the length of one edge of the base of the pyramid; you pick two points.

Specify first endpoint of edge: *Specify a point*
Specify second endpoint of edge: *Specify a point*

### Sides

Specifies the number of sides for the pyramid. You can enter a number from 3 to 32.

Specify number of sides <default>: *Specify a diameter or press* ENTER *to specify the default value*

Initially, the number of sides for the pyramid is set to 4. During a drawing session, the default value for the number of sides is always the previously entered value for the number of sides.

### Inscribed

Specifies that the base of the pyramid is inscribed within (drawn within) the base radius of the pyramid.

### Circumscribed

Specifies that the pyramid is circumscribed around (drawn around) the base radius of the pyramid.

### 2Point

Specifies that the height of the pyramid is the distance between two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

### Axis Endpoint

Specifies the endpoint location for the pyramid axis. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the pyramid.

Specify axis endpoint: *Specify a point*

### Top Radius

Specifies the top radius of the pyramid, creating a pyramid frustum.

Specify top radius: *Enter a value*

Initially, the default top radius is not set to any value. During a drawing session, the default value for the top radius is always the previously entered top radius value for any solid primitive.

Specify height or [2Point (Two Points)/Axis Endpoint] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

### 2Point

Specifies that the height of the pyramid is the distance between two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

### Axis Endpoint

Specifies the endpoint location for the pyramid axis. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the pyramid.

Specify axis endpoint: *Specify a point*

# Q Commands

# 17

**In this chapter**

- QCCLOSE
- QDIM
- QLEADER
- QNEW
- QSAVE
- QSELECT
- QTEXT
- QUICKCALC
- QUICKCUI
- QUIT

# QCCLOSE

## Quick Reference

Closes QuickCalc

**qcclose**

Closes the QuickCalc calculator.

# QDIM

## Quick Reference

Quickly creates a dimension

Dimension 
Dimension ➤ Quick DimensionAt the Command prompt, enter qdim.
**qdim**

Use QDIM to quickly create or edit a series of dimensions. The command is particularly useful for creating a series of baseline or continued dimensions, or for dimensioning a series of circles and arcs.

Select geometry to dimension: *Select the objects you want to dimension or the dimensions you want to edit and press* ENTER
Specify dimension line position, or [Continuous (page 1144)/Staggered (page 1144)/Baseline (page 1144)/Ordinate (page 1144)/Radius (page 1144)/Diameter (page 1144)/datumPoint (page 1145)/Edit (page 1145)/seTtings (page 1145)] <*current*>: *Enter an option or press* ENTER
**Continuous** Creates a series of continued dimensions.

**Staggered** Creates a series of staggered dimensions.

**Baseline** Creates a series of baseline dimensions.

**Ordinate** Creates a series of ordinate dimensions.

**Radius** Creates a series of radius dimensions.

**Diameter** Creates a series of diameter dimensions.

**Datum Point** Sets a new datum point for baseline and ordinate dimensions.

Select new datum point: *Specify a point*

The program returns to the previous prompt.

**Edit** Edits a series of dimensions. You are prompted to add or remove points from existing dimensions.

Indicate dimension point to remove, or [Add/eXit] <eXit>: *Specify a point, enter* **a***, or press* ENTER *to return to the previous prompt*

**Settings** Sets the default object snap for specifying extension line origins. The following prompt is displayed:

Associative dimension priority [Endpoint/Intersection]

The program returns to the previous prompt.

# QLEADER

## Quick Reference

Creates a leader and leader annotation

**qleader**

It is recommended that you use the workflow available through the MLEADER (page 838) command to create leader objects. For more information about multileader objects, see Create and Modify Leaders.

Use QLEADER to quickly create leaders and leader annotation. You can use the Leader Settings dialog box (page 1147) to customize the command so that it prompts you for the number of leader points and the annotation type suited to your drawing needs. You can use QLEADER to

- Specify leader annotation and annotation format
- Set the location where leaders attach to multiline text annotation
- Limit the number of leader points
- Constrain the angle of the first and second leader segments

If associative dimensioning is turned on with DIMASSOC, the leader start point can be associated with a location on an object. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the text or feature control frame remains in place.

Specify first leader point (page 1146), or [Settings (page 1147)] <Settings>: *Specify the first leader point, or press* ENTER *to specify leader settings*

### First Leader Point

Specify next point: *Specify the next leader point*
Specify next point: *Specify the next leader point, or press* ENTER *to specify the leader annotation*

The Number of Points setting on the Leader Line & Arrow tab of the Leader Settings dialog box (page 1147) determines the number of leader points you are prompted to specify.

The prompt that is displayed next depends on settings selected on the Annotation tab in the Leader Settings dialog box.

If Mtext and Prompt for Width are selected on the Annotation tab, the following prompts are displayed:

Specify the width <*current*>: *Specify the multiline text width by creating a text boundary or entering a value*

If you set the text width value to 0.00, the width of the multiline text is unlimited.

Enter first line of annotation text: *Enter the first line of text*

Press ENTER once to enter another line of text, or press ENTER again to complete the command.

If Copy an Object is selected on the Annotation tab, the following prompt is displayed:

Select an object to copy: *Select a text object, block reference, or tolerance object (feature control frame)*

The object is attached to the leader.

If Tolerance is selected on the Annotation tab, the Geometric Tolerance dialog box (page 1437) is displayed. Use the dialog box to create the tolerance feature control frame. When you choose OK, the feature control frame is attached o the leader.

If Block Reference is selected on the Annotation tab, the following prompts are displayed:

Enter block name or [?]: *Enter the name, or enter* **?** *to display a list of blocks defined in the drawing*
Specify insertion point or [Scale/X/Y/X/Rotate/PScale/PX/PY/PZ/PRotate]: *Specify the block insertion point or enter an option*

For a description of the insertion options, see -INSERT (page 672).

If None is selected on the Annotation tab, no annotation prompts are displayed.

### Settings

Displays the Leader Settings dialog box (page 1147).

# Leader Settings Dialog Box

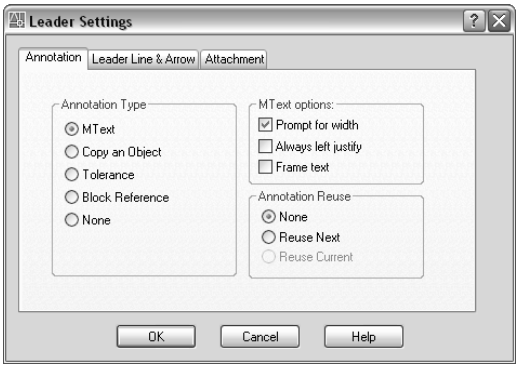
## Quick Reference

### qleader

Customizes the QLEADER command and sets properties for leaders and leader annotations.

### Annotation Tab (Leader Settings Dialog Box)

Sets the leader annotation type, specifies multiline text options, and indicates whether you want to reuse the annotation.




### Annotation Type

Sets the leader annotation type. The type you select changes the QLEADER leader annotation prompt.

**MText** Prompts you to create multiline text (mtext) annotation.

**Copy an Object** Prompts you to copy a multiline text, single-line text, tolerance, or block reference object and connects the copy to the end of the leader line. The copy is associated with the leader line, meaning that if the copied object moves, the end of the leader line moves with it. The display of the hook line depends on the object copied.

**Tolerance** Displays the Tolerance dialog box, which you can use to create a feature control frame to attach to the leader.

**Block Reference** Prompts you to insert a block reference. The block reference is inserted at an offset from the end of the leader line and is associated to the leader line, meaning that if the block moves, the end of the leader line moves with it. No hook line is displayed.

**None** Creates a leader with no annotation.

### MText Options

Sets multiline text options. The options are available only when the multiline text annotation type is selected.

**Prompt for Width** Prompts you to specify the width of the multiline text annotation.

**Always Left Justify** Left-justifies the multiline text annotation, regardless of leader location.

**Frame Text** Places a frame around multiline text annotation.

### Annotation Reuse

Sets options for reusing leader annotation.

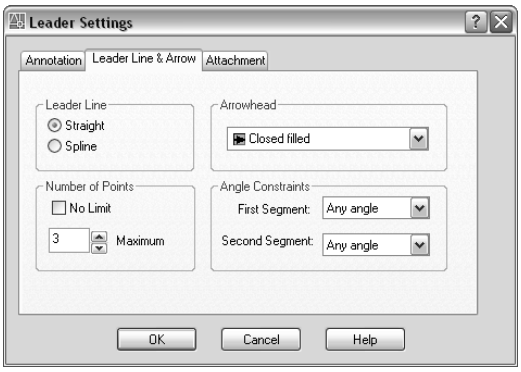
**None** Does not reuse leader annotation.

**Reuse Next** Reuses the next annotation you create for all subsequent leaders.

**Reuse Current** Reuses current annotation. This option is automatically selected when you reuse annotation after selecting Reuse Next.

### Leader Line & Arrow Tab (Leader Settings Dialog Box)

Sets the leader line and arrowhead format.




### Leader Line

Sets the leader line format.

**Straight** Creates straight-line segments between the points you specify.

**Spline** Creates a spline object using the leader points you specify as control points.

### Arrowhead

Defines the leader arrowhead. Select an arrowhead from the Arrowhead list. The arrowheads are the same ones that are available for dimension lines. See *DIMSTYLE*. If you select User Arrow, a list of blocks in the drawing is displayed. Select one of the blocks to use it as a leader arrowhead.

### Number of Points

Sets the number of leader points that QLEADER prompts you to specify before prompting for the leader annotation. For example, if you set the points to 3, QLEADER automatically prompts you to specify the annotation after you specify two leader points. Set the number to one more than the number of leader segments you want to create.

If you set the option to No Limit, QLEADER prompts for leader points until you press ENTER.

### Angle Constraints

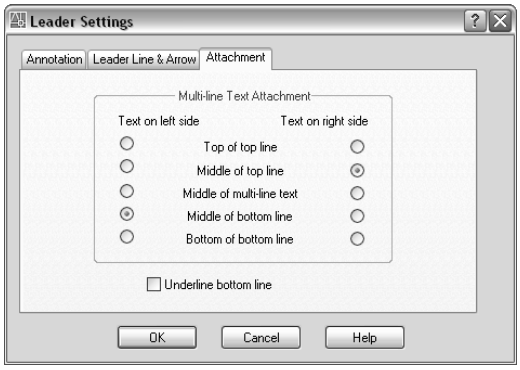
Sets angle constraints for the first and second leader lines.

**First Segment** Sets the angle of the first leader segment.

**Second Segment** Sets the angle of the second leader segment.

### Attachment Tab (Leader Settings Dialog Box)

Sets the attachment location for leader lines and multiline text annotation. This tab is available only when Mtext is selected on the Annotation tab.




**Top of Top Line** Attaches the leader line at the top of the top multiline text line.

**Middle of Top Line** Attaches the leader line at the middle of the top multiline text line.

**Middle of Multiline Text** Attaches the leader line at the middle of the multiline text.

**Middle of Bottom Line** Attaches the leader line at the middle of the bottom multiline text line.

**Bottom of Bottom Line** Attaches the leader line at the bottom of the bottom multiline text line.

**Underline Bottom Line** Underlines the bottom multiline text line.

# QNEW

## Quick Reference

Starts a new drawing with the option of using a default drawing template file

Standard 
**qnew**

QNEW starts a new drawing from the current default drawing template file and folder path specified in the Options dialog box on the Files tab. You can set the default drawing template file to any drawing template file or to *None*.

When a default drawing template file is set to *None* or is not specified, QNEW displays the Select Template File dialog box (a standard file selection dialog box (page 931)).

The behavior of the QNEW command is determined by the *STARTUP* system variable.

- *1:* Displays the Create New Drawing dialog box (page 911).
- *0:* Displays the Select Template dialog box (a standard file selection dialog box (page 931)) or starts the new drawing using the default drawing template file.

If the system variable, FILEDIA, is set to 0 instead of 1, a command prompt (page 917) is displayed. If you set *FILEDIA* to 0, this prompt is displayed regardless of the Startup setting.

# QSAVE

## Quick Reference

Saves the current drawing using the file format specified in the Options dialog box

Standard 
File ➤ SaveAt the Command prompt, enter qsave.
**qsave**

The QSAVE command is equivalent to clicking Save on the File menu.

If the drawing is named, the program saves the drawing using the file format specified on the Open and Save tab of the Options dialog box and does not request a file name. If the drawing is unnamed, the Save Drawing As dialog

box (see *SAVEAS*) is displayed and the drawing is saved with the file name and format you specify.

If the drawing is read-only, use the SAVEAS command to save the changed file under a different name.

# QSELECT

## Quick Reference

Creates a selection set based on filtering criteria

Tools ➤ Quick SelectAt the Command prompt, enter qselect.
End any active commands, right-click in the drawing area, and choose Quick Select.
**qselect**

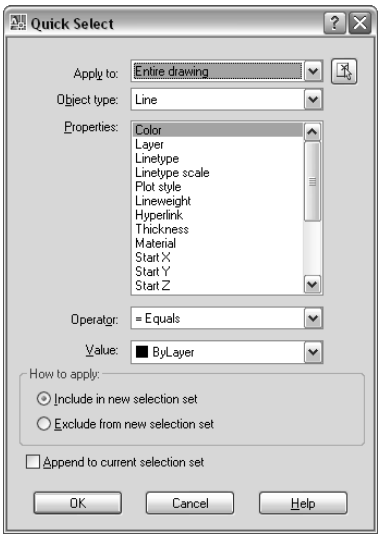
The Quick Select dialog box (page 1152) is displayed.

# Quick Select Dialog Box

## Quick Reference

Tools ➤ Quick SelectAt the Command prompt, enter qselect.
End any active commands, right-click in the drawing area, and choose Quick Select.
**qselect**

Specifies the filtering criteria and how you want to create the selection set from that criteria.




**Apply To** Applies the filtering criteria to the entire drawing or to the current selection set (if one exists). To select a group of objects to which you want to apply the filtering criteria, use the Select Objects button. When you have finished selecting objects, press ENTER to redisplay the dialog box. Apply To is set to Current Selection.

If Append to Current Selection Set is selected, the filtering criteria is applied to the entire drawing.

**Select Objects** Temporarily closes the Quick Select dialog box so that you can select the objects to which you want to apply the filter criteria. Press ENTER to return to the Quick Select dialog box. The Apply To box is changed to show Current Selection. The Select Objects button is available only when you select Include In New Selection Set and clear Append to Current Selection Set.

**Object Type** Specifies the type of objects to include in the filtering criteria. If the filtering criteria are being applied to the entire drawing, the Object Type list includes all object types, including custom. Otherwise, the list includes only the object types of the selected objects.

If an application such as Autodesk Map was used to add a feature classification to an object, you can select a classification.

**Properties** Specifies the object property for the filter. This list includes all searchable properties for the selected object type. The property you select determines the options available in Operator and Value.

If an application such as Autodesk® Map™ was used to add a feature classification to an object, you can select a classification property.

**Operator** Controls the range of the filter. Depending on the selected property, options may include Equals, Not Equal To, Greater Than, Less Than, and *Wildcard Match. Greater Than and Less Than are not available for some properties. *Wildcard Match is available only for text fields that can be edited.

For information about the available wild-card characters, see the table in "Filter and Sort the List of Layers" in the *User's Guide*.

**Value** Specifies the property value for the filter. If known values for the selected property are available, Value becomes a list in which you can choose a value. Otherwise, enter a value.

**How to Apply** Specifies whether you want the new selection set to include or exclude objects that match the specified filtering criteria. Select Include in New Selection Set to create a new selection set composed only of objects that match the filtering criteria. Select Exclude from New Selection Set to create a new selection set composed only of objects that do not match the filtering criteria.

**Append to Current Selection Set** Specifies whether the selection set created by QSELECT replaces or is appended to the current selection set.

---

**NOTE** QSELECT supports custom objects (objects created by another application) and their properties. If a custom object uses properties other than AutoCAD properties, the custom object's source application must be running in order for the properties to be available to QSELECT.

---

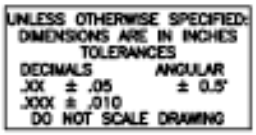
# QTEXT

## Quick Reference

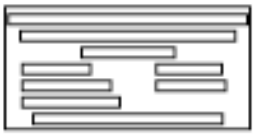
Controls the display and plotting of text and attribute objects

**qtext** (or '**qtext** for transparent use)
Enter mode [ON/OFF] <*current*>: *Enter* **on** *or* **off**, *or press* ENTER

When QTEXT (Quick Text) is on, each text and attribute object is displayed as a bounding box around the text object. Turning QTEXT mode on reduces the time it takes the program to redraw and regenerate drawings that contain many text objects.



QTEXT off


QTEXT on

# QUICKCALC

## Quick Reference

Opens the QuickCalc calculator

Standard 

Tools ➤ Palettes ➤ QuickCalcAt the Command prompt, enter quickcalc.
Right-click and click QuickCalc.

**quickcalc**

Displays the QuickCalc calculator (page 1155).

# QuickCalc Calculator

## Quick Reference

Performs a full range of mathematical, scientific, and geometric calculations, creates and uses variables, and converts units of measurement.

The QuickCalc calculator contains the following areas:

- Toolbar (page 1156)
- History Area (page 1156)
- Input Box (page 1157)
- More/Less Button (page 1157)
- Number Pad (page 1157)

- Scientific Area (page 1159)
- Units Conversion Area (page 1160)
- Variables Area (page 1161)

### Toolbar

Performs quick calculations of common functions.

**Clear**  Clears the Input box.

**Clear History**  Clears the history area.

**Paste Value to Command Line**  Pastes the value in the Input box at the command prompt. When QuickCalc is used transparently during a command, this button is replaced by the Apply button at the bottom of the calculator.

**Get Coordinates**  Calculates the coordinates of a point location that you have clicked in the drawing.

**Distance Between Two Points**  Calculates the distance between two point locations that you have clicked on an object.

**Angle of Line Defined by Two Points**  Calculates the angle of two point locations that you have clicked on an object.

**Two Lines Defined by Four Points**  Calculates the intersection of four point locations that you have clicked on an object.

**Help**  Displays Help for QuickCalc.

### History Area

Displays a running list of previously evaluated expressions. The History area shortcut menu provides several options, including copying a selected expression to the Clipboard.

### Input Box

Provides a box where you enter and retrieve expressions. When you click the = (equal) button or press ENTER, QuickCalc evaluates an expression and displays the results.

### More/Less Button

Hides or Displays all QuickCalc function areas. You can also right-click the button to select the individual function areas to hide or display.

### Number Pad

Provides a standard calculator keypad where you enter numbers and symbols for arithmetic expressions. Enter values and expressions, and then click the equal (=) sign to evaluate the expression. The following table describes additional controls found on the Number Pad.

| Control | Description |
|---|---|
| C (Clear) | Clears any entry in the Input box and resets its value to 0. |
| <-- (Back-space) | Moves the cursor one space to the left in the Input box, removing one decimal place or character from the display. |
| sqt (Square Root) | Obtains the square root of a value. |
| 1/X (In-verse) | Inverts any number or expression entered in the Input box. |
| x^2 (X to the Power of 2) | Squares a value. |
| x^3 (X to the | Raises any number or expression entered in the Input box to the power of 3. |

| Control | Description |
|---|---|
| Power of 3) | |
| x^y (X to the Power of Y) | Raises a number or expression entered in the Input box to a specified power. |
| pi | Enters pi to 14 decimal places in the Input box. |
| ( (Open Parenthesis) ) (Close Parenthesis) | When combined in pairs, groups a portion of the expression. Items contained in a parenthetical grouping are evaluated before the remainder of the expression. |
| = (Equal) | Evaluates the expression currently entered in the Input box. |
| MS (Store in Memory) | Stores the current value in the QuickCalc memory. |
| M+ (Add to Value Stored in Memory) | Adds the current value to the value stored in the QuickCalc memory. |
| MR (Restore Memory Value) | If a value is currently stored in the QuickCalc memory, the value is restored to the Input box. |
| MC (Clear Memory) | Clears the value currently stored in the QuickCalc memory. |

### Scientific Area

Evaluates trigonometric, logarithmic, exponential, and other expressions commonly associated with scientific and engineering applications. The following table describes the controls in the Scientific Area.

| Control | Description |
|---|---|
| sin (Sine) | Specifies the sine of the angle in the Input box. |
| cos (Cosine) | Specifies the cosine of the angle in the Input box. |
| tang (Tangent) | Specifies the tangent of the angle in the Input box. |
| Log (Base - 10 Log) | Specifies the log of the value in the Input box. |
| 10^x (Base - 10 Exponent) | Specifies the base-10 exponent of the value in the Input box. |
| asin (Arcsine) | Specifies the arcsine of the number in the Input box. The number must be between -1 and 1. |
| acos (Arccosine) | Specifies the arccosine of the number in the Input box. The number must be between -1 and 1. |
| atan (Arctangent) | Speifies the arctangent of the number in the Input box. |
| ln (Natural Log) | Specifies the natural log of the number in the Input box. |

| Control | Description |
|---|---|
| e^x (Natural Exponent) | Specifies the natural exponent of the number currently specified in the Input box. |
| r2d (Convert Radians to Degrees) | Converts angles in radians to degrees; for example, r2d (pi) converts the value of pi to 180 degrees. |
| d2r (Convert Degrees to Radians) | Converts angles in degrees to radians; for example, d2r (180) converts 180 degrees to radians and returns the value of pi. |
| abs (Absolute Value) | Returns the absolute value of the number in the Input box. |
| rnd (Round) | Rounds the number in the Input box to the nearest integer. |
| trunc (Truncate) | Returns the integer portion of the number in the Input box. |

### Units Conversion Area

Converts units of measurement from one unit type to another unit type. The units conversion area accepts only decimal values without units.

**Units Type** Select length, area, volume, and angular values from a list.

**Convert From** Lists the units of measurement from which to convert.

**Convert To** Lists the units of measurement to which to convert.

**Value to Convert** Provides a box to enter a value to convert.

**Converted Value** Converts the units entered and displays the converted value.

**Calculator Icon** Returns the converted value to the Input box.

### Variables Area

Provides access to predefined constants and functions. You can use the Variables area to define and store additional constants and functions.

**Variables Tree** Stores predefined shortcut functions and user-defined variables.

Shortcut functions are common expressions that combine a function with an object snap. The following table describes the predefined shortcut functions in the list.

| Shortcut Function | Shortcut For | Description |
|---|---|---|
| dee | dist(end,end) | Distance between two endpoints |
| ille | ill(end,end,end,end) | Intersection of two lines defined by four endpoints |
| mee | (end+end)/2 | Midpoint between two endpoints |
| nee | nor(end,end) | Unit vector in the *XY* plane and normal to two endpoints |
| rad | rad | Radius of a selected circle, arc, or polyline arc |
| vee | vec(end,end) | Vector from two endpoints |
| vee1 | vec1(end,end) | Unit vector from two endpoints |

**New Variable Button** Opens the Variable Definition dialog box (page 1162).

**Edit Variable Button** Opens the Variable Definition dialog box (page 1162) so you can make changes to the selected variable.

**Delete Variable Button** Deletes the selected variable.

**Calculator Button** Returns the selected variable to the Input box.

# Variable Definition Dialog Box

## Quick Reference

Defines a variable to store in the variables tree.

### Variable Type

Specifies the type of variable.

**Constant** Stores the new variable as a constant.

**Function** Stores the new variable as a function.

### Variable Properties

Defines the properties of a new variable.

**Name** Stores the name of the variable. Names of constants cannot include spaces.

**Group With** Stores the named variable in the selected category. If New is selected, the Category Definition dialog box (page 1163) opens, where you can define a new category for the variable.

**Value or Expression** Stores the value or expression for the variable.

**Description** Stores a description for the variable.

**OK** Closes the dialog box and applies the current settings to the variable.

**Cancel** Closes the dialog box without applying the current settings.

**Help** Displays Help.

## Category Definition Dialog Box

### Quick Reference

Defines a category to organize the variables in the variables tree.

#### Category Properties

Defines the properties of the new category.

**Name** Stores the name of the category.

**Description** Stores a description for the category.

**OK** Closes the dialog box and applies the current settings to the variable.

**Cancel** Closes the dialog box without applying the current settings.

**Help** Displays Help.

# QUICKCUI

### Quick Reference

Displays the Customize User Interface dialog box in a collapsed state

#### quickcui

The Customize User Interface dialog box (page 276) is displayed in a collapsed state. Only the Customizations In *<filename>* and Command List panes are displayed. The Customizations In *<filename>* pane is shown collapsed and the Command List pane is shown expanded.

For information about customizing the different user interface elements found in the Customize User Interface dialog box, see Customize the User Interface in the *Customization Guide*.

# QUIT

## Quick Reference

Exits the program

File ➤ ExitAt the Command prompt, enter quit.
**quit**

Quits the program if there have been no changes since the drawing was last saved. If the drawing has been modified, you are prompted to save or discard the changes before quitting.

You can quit a file that has been opened in read-only mode if you have made no modifications or if you are willing to discard them. To save modifications to a read-only drawing, use the *SAVEAS* command to save the drawing under another name.

# R Commands

# 18

**In this chapter**

- RENDERPRESETS
- RENDERWIN
- RENDSCR
- REPLAY
- RESETBLOCK
- RESUME
- REVCLOUD
- REVOLVE
- REVSURF
- RMAT
- ROTATE
- ROTATE3D
- RPREF
- RPREFCLOSE
- RSCRIPT
- RULESURF

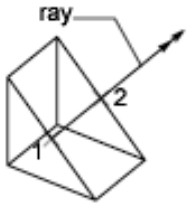
# RAY

## Quick Reference

Creates a semi-infinite line

Draw ➤ RayAt the Command prompt, enter ray.
**ray**
Specify start point: *Specify a point (1)*
Specify through point: *Specify a point for the ray to pass through (2)*

The ray is extended to the edge of the display in the direction defined by the starting point and the through point. The prompt for a through point is redisplayed so you can create multiple rays. Press ENTER to end the command.




# RECOVER

## Quick Reference

Repairs a damaged drawing

File ➤ Drawing Utilities ➤ RecoverAt the Command prompt, enter recover.
**recover**

In the Select File dialog box (a standard file selection dialog box (page 931)), enter the drawing file name or select the damaged drawing file. Results are displayed in the text window.

When *FILEDIA* is set to 0 (zero), RECOVER displays the following command prompt.

Enter name of drawing file to recover:

Enter ~ (tilde) at the prompt to ignore FILEDIA and display the Select File dialog box.

**NOTE** The RECOVER command recovers or audits DWG, DWT, and DWS files. Performing a recover on a DXF file will only open the file.

# RECOVERALL

## Quick Reference

Repairs a damaged drawing and xrefs

File ➤ Drawing Utilities ➤ Recover Drawing and Xrefs
**recoverall**

In the Select File dialog box (a standard file selection dialog box (page 931)), enter the drawing file name or select the damaged drawing file.

The selected drawing file and all attached xrefs, including all nested xrefs, are opened, repaired, resaved, and closed.

- Drawing files are saved in the current drawing file format.
- Copies of the original drawing files are saved as BAK files.
- If the object enabler is present, custom objects are updated.

Results are displayed in the Drawing Recovery Log window. Each drawing file checked includes a Drawing Recovery Log that can be expanded or collapsed. The entire log can be copied to the Windows clipboard with the Copy to Clipboard button.

When *FILEDIA* is set to 0 (zero), RECOVERALL displays the following command prompt.

Enter name of drawing file to recover:

Enter ~ (tilde) at the prompt to ignore FILEDIA and display the Select File dialog box.

**NOTE** The RECOVERALL command recovers or audits DWG, DWT, and DWS files.

# RECTANG

## Quick Reference

Draws a rectangular polyline

Draw

Draw ➤ RectangleAt the Command prompt, enter rectang.

**rectang** or **rectangle**

Current settings: Rotation = 0

Specify first corner point (page 1169) or [Chamfer (page 1170)/Elevation (page 1170)/Fillet (page 1170)/Thickness (page 1171)/Width (page 1171)]: *Specify a point or enter an option*

### First Corner Point

Specifies a corner point of the rectangle.

Specify other corner point or [Area/Dimensions/Rotation]: *Specify a point or enter an option*

**Other Corner Point**


Creates a rectangle using the specified points as diagonally opposite corners.

**Area** Creates a rectangle using the area and either a length or a width. If the Chamfer or Fillet option is active, the area includes the effect of the chamfers or fillets on the corners of the rectangle.

Enter area of rectangle in current units <100>: *Enter a positive value*

Calculate rectangle dimensions based on [Length/Width] <Length>: *Enter* **L** *or* **w**

Enter rectangle length <10>: *Enter a non-zero value*

or

Enter rectangle width <10>: *Enter a non-zero value*

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

**Dimensions** Creates a rectangle using length and width values.

Specify length for rectangles <0.0000> *Enter a non-zero value*

Specify width for rectangles <0.0000> *Enter a non-zero value*

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

**Rotation** Creates a rectangle at a specified rotation angle.

Specify rotation angle or [Points] <0> Specify an angle by entering a value, specifying a point, or entering **p** and specifying two points

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

### Chamfer

Sets the chamfer distances for the rectangle.

Specify first chamfer distance for rectangles <*current*>: *Specify a distance or press* ENTER
Specify second chamfer distance for rectangles <*current*>: *Specify a distance or press* ENTER

The values become the current chamfer distances for subsequent RECTANG commands.

### Elevation

Specifies the elevation of the rectangle.

Specify the elevation for rectangles <*current*>: *Specify a distance or press* ENTER

The value becomes the current elevation for subsequent RECTANG commands.

### Fillet

Specifies the fillet radius of the rectangle.

Specify fillet radius for rectangles <*current*>: *Specify a distance or press* ENTER

The value becomes the current fillet radius for subsequent RECTANG commands.

#### Thickness

Specifies the thickness of the rectangle.

Specify thickness for rectangles <*current*>: *Specify a distance or press* ENTER

The value becomes the current thickness for subsequent RECTANG commands.

#### Width

Specifies the polyline width of the rectangle to be drawn.

Specify line width for rectangles <*current*>: *Specify a distance or press* ENTER

The value becomes the current polyline width for subsequent RECTANG commands.

# REDEFINE

## Quick Reference

Restores AutoCAD internal commands overridden by UNDEFINE

**redefine**
Enter command name: *Enter the name of an AutoCAD command turned off by the UNDEFINE command*

If a command has been undefined, you can still use it if you precede the command name with a period.

# REDO

## Quick Reference

Reverses the effects of previous UNDO or U command

Standard 
Edit ➤ RedoAt the Command prompt, enter mredo.
With no commands active and no objects selected, right-click in the drawing area and choose Redo.

**redo**

REDO reverses the effects of a single *UNDO* or *U* command. REDO must immediately follow the U or UNDO command.

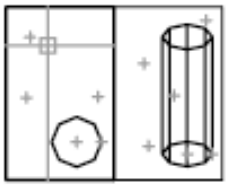
# REDRAW

## Quick Reference

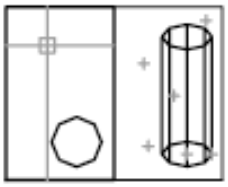
Refreshes the display in the current viewport

**redraw** (or '**redraw** for transparent use)

When *BLIPMODE* is on, marker blips left by editing commands are removed from the current viewport.



before REDRAW



after REDRAW

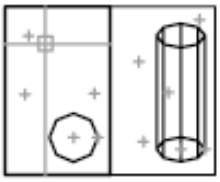
# REDRAWALL

## Quick Reference

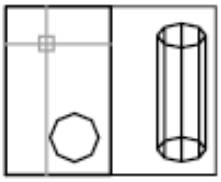
Refreshes the display in all viewports

View ➤ RedrawAt the Command prompt, enter redrawall.
**redrawall** (or '**redrawall** for transparent use)

When *BLIPMODE* is on, marker blips left by editing commands are removed from all viewports.



before REDRAWALL



after REDRAWALL

# REFCLOSE

## Quick Reference

Saves back or discards changes made during in-place editing of a reference (an xref or a block)

Tools ➤ Xref and Block In-Place Editing ➤ Save Reference Edits, or Xref and Block In-Place Editing ➤ Close Reference

With no objects selected during in-place reference editing, right-click in the drawing area and choose Close Refedit Session.

**refclose**

Enter option [Save (page 1173)/Discard reference changes (page 1173)] <Save>:

**Save** Saves back to the xref source drawing or to the block definition in the current drawing all changes made to objects in the working set. If you remove an object from the working set and save changes, the object is deleted from the reference and added to the current drawing. The Save Back Changes to Reference button on the Refedit toolbar automatically saves reference editing changes.

**NOTE** If the file format of the xref source drawing is AutoCAD Release 14 or earlier, the file is saved in AutoCAD 2007 file format. The file format is not changed for xref source drawings in AutoCAD 2000 and 2004 format.

**Discard Reference Changes** Discards the working set; the source drawing or block definition is returned to its original state. Any changes you make to objects in the current drawing (not in the xref or block) are not discarded. If you delete any object that is not in the working set, the object is not restored even if you choose to discard changes. The Discard Changes to Reference

button on the Refedit toolbar automatically discards reference editing changes without using REFCLOSE.

If you save or discard changes with REFCLOSE, you can still use the *UNDO* command to return to the reference editing session. If you have made unwanted changes to an xref and already saved back the changes, use UNDO to undo the unwanted changes; then use REFCLOSE to save back changes and restore the xref to its original state.

---

**NOTE** When you edit and save xrefs in place in a drawing, the preview image for the original reference drawing is no longer available unless you open and save the drawing again.

---

# REFEDIT

### Quick Reference

Selects an external reference or block reference for editing

Refedit
Tools ➤ Xref and Block In-place Editing ➤ Edit Reference In-Place
**refedit**
Select reference: *Select an xref or a block in the current drawing*

The Reference Edit dialog box (page 1174) is displayed.

If you enter **-refedit** at the command prompt, options are displayed at the command prompt (page 1177).

## Reference Edit Dialog Box

### Quick Reference

Refedit
Tools ➤ Xref and Block In-place Editing ➤ Edit Reference In-Place

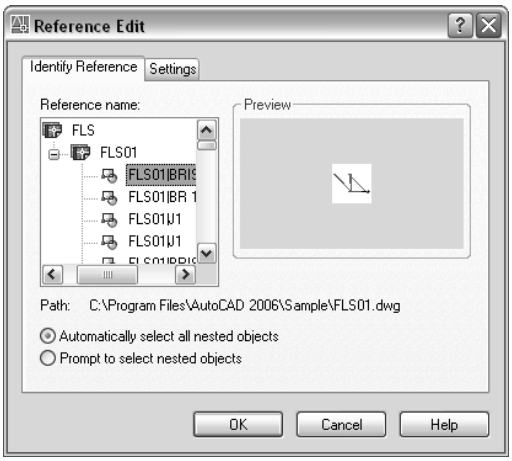
**refedit**

Specifies the reference to edit. To select a reference to edit, select an object in the reference. If you select an object that is part of one or more nested references, the nested references are displayed in the dialog box.

- Identify Reference (page 1175)
- Settings (page 1176)

### Identify Reference Tab (Reference Edit Dialog Box)

Provides visual aids for identifying the reference to edit and controls how the reference is selected.




**Reference Name** Displays the reference selected for in-place editing and any references nested within the selected reference.

Nested references are displayed only if the selected object is part of a nested reference. If multiple references are displayed, choose a specific xref or block to modify. Only one reference can be edited in place at a time.

**Preview** Displays a preview image of the currently selected reference.

The preview image displays the reference as it was last saved in the drawing. The reference preview image is not updated when changes are saved back to the reference.

**Path** Displays the file location of the selected reference. If the selected reference is a block, no path is displayed.

**Automatically Select All Nested Objects** Controls whether nested objects are included automatically in the reference editing session.

If this option is checked, all the objects in the selected reference will be automatically included in the reference editing session.

**Prompt to Select Nested Objects** Controls whether nested objects must be selected individually in the reference editing session.

If this option is checked, after you close the Reference Edit dialog box and enter the reference edit state, you are prompted to select the specific objects in the reference that you want to edit.

Select nested objects: *Select objects within the reference that you want to edit*

### Settings Tab (Reference Edit Dialog Box)

Provides options for editing references.

**Create Unique Layer, Style, and Block Names** Controls whether layers and other named objects extracted from the reference are uniquely altered.

If selected, named objects in xrefs are altered (names are prefixed with $#$), similar to the way they are altered when you bind xrefs. If cleared, the names of layers and other named objects remain the same as in the reference drawing. Named objects that are not altered to make them unique assume the properties of those in the current host drawing that share the same name.

**Display Attribute Definitions for Editing** Controls whether all variable attribute definitions in block references are extracted and displayed during reference editing.

If Display Attribute Definitions for Editing is selected, the attributes (except constant attributes) are made invisible, and the attribute definitions are available for editing along with the selected reference geometry. When changes are saved back to the block reference, the attributes of the original reference remain unchanged. The new or altered attribute definitions affect only subsequent insertions of the block; the attributes in existing block instances are not affected. Xrefs and block references without definitions are not affected by this option.

**Lock Objects Not in Working Set** Locks all objects not in the working set. This prevents you from accidentally selecting and editing objects in the host drawing while in a reference editing state.

The behavior of locked objects is similar to objects on a locked layer. If you try to edit locked objects, they are filtered from the selection set.

# -REFEDIT

## Quick Reference

If you enter **-refedit** at the command prompt, the following REFEDIT command prompts are displayed.

Select reference: *Select an xref or block in the current drawing*
Select nesting level [OK (page 1177)/Next (page 1177)] <Next>: *Enter an option or press* ENTER
**OK** Accepts the currently highlighted reference for in-place reference editing.

Enter object selection method [All/Nested]<All>: *Enter* **a**, *enter* **n**, *or press* ENTER.

If you enter **a,** all the objects in the selected reference will be automatically included in the reference editing session. If you enter **n**, after you close the Reference Edit dialog box and enter the reference edit state, you are prompted to select the specific objects in the reference that you want to edit.

Select nested objects: *Select objects within the reference that you want to edit*

Display attribute definitions [Yes/No] <No>: *Enter* **y**, *enter* **n**, *or press* ENTER *when editing a block reference*

Use REFCLOSE or the Refedit toolbar to end reference editing session.

If you are editing a block reference with attributes, you can enter **y** to display the attribute definitions and make them available for editing. The attributes are made invisible, and the attribute definitions are available for editing along with the selected reference geometry. When changes are saved back to the block reference, the attributes of the original reference remain unchanged. The new or altered attribute definitions affect only subsequent insertions of the block; the attributes in existing block instances are not affected.

**Next** Advances through the reference and nested references available for selection. The currently selected reference is highlighted.

A working set is formed with the objects you have selected for editing. The working set includes objects that can be saved back to update the xref or block definition. When you save back changes, changes made to the objects in the reference file are saved without actually opening the reference drawing or recreating the block. The working set is visually distinct from the rest of the current drawing: all objects in the current drawing, except objects in the working set, appear faded. The *XFADECTL* system variable controls the fading of objects while you edit a reference in place.

You can select objects in xrefs for editing even if they are on a locked layer in the reference file. When a reference object is part of the working set, you can

unlock the object's layer and make changes to the object. Only the changes made to the object are saved back to the reference file; the xref layer remains locked in the reference file.

**NOTE** Objects outside of the working set are not faded unless the visual style is set to 2D Wireframe during in-place reference editing.

# REFSET

## Quick Reference

Adds or removes objects from a working set during in-place editing of a reference (an xref or a block)

Tools ➤ Xref and Block In-place Editing ➤ Add to Working Set, or Xref and Block In-Place Editing ➤ Remove from Working Set
**refset**
Transfer objects between the Refedit working set and host drawing...
Enter an option [Add (page 1178)/Remove (page 1178)] <Add>: *Enter an option or press* ENTER

Objects that are part of the working set are visually distinct from other objects in the current drawing. All objects in the current drawing, except objects in the working set, appear faded.

**Add** Adds objects to the working set.

Select objects: *Select the objects you want to add*

An object that is part of the working set is added to the reference when changes are saved back, and the object is removed from the current drawing. The Add to Workset button on the Refedit toolbar prompts you to add objects to the working set without using REFCLOSE.

**Remove** Removes objects from the working set.

Select objects: *Select the objects you want to remove*

An object that is removed from the working set is removed from the reference when changes are saved back; the object is also removed from the current drawing. The Remove from Workset button on the Refedit toolbar prompts you to remove objects from the working set without using *REFCLOSE*.

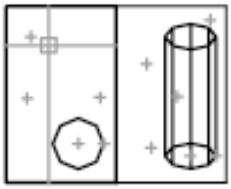
# REGEN

## Quick Reference

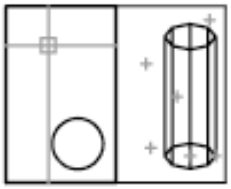
Regenerates the entire drawing from the current viewport

View ➤ RegenAt the Command prompt, enter regen.
**regen**

REGEN regenerates the entire drawing and recomputes the screen coordinates for all objects in the current viewport. It also reindexes the drawing database for optimum display and object selection performance.



before REGEN



after REGEN

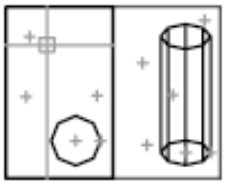
# REGENALL

## Quick Reference

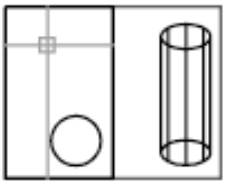
Regenerates the drawing and refreshes all viewports

View ➤ Regen AllAt the Command prompt, enter regenall.
**regenall**

REGENALL regenerates the entire drawing and recomputes the screen coordinates for all objects in all viewports. It also reindexes the drawing database for optimum display and object selection performance.



before REGENALL



after REGENALL

# REGENAUTO

## Quick Reference

Controls automatic regeneration of a drawing

**regenauto** (or '**regenauto** for transparent use)
Enter mode [ON (page 1180)/OFF (page 1180)] <*current*>: *Enter* **on** *or* **off**, *or press* ENTER

**On** Regenerates the drawing immediately if any suppressed regenerations exist in the queue and continues to regenerate automatically whenever you perform an action that requires regeneration.

**Off** Inhibits regeneration of the drawing until you use the *REGEN* or *REGENALL* command, or set REGENAUTO to on.

If you perform an action that requires a regeneration and that action is irrevocable (such as thawing layers), the following message is displayed:

Regen queued

If you perform an action that requires a regeneration and that action is revocable, the following message is displayed:

About to regen—proceed?

If you click OK, the drawing is regenerated. If you click Cancel, the last action is cancelled and the drawing is not regenerated.

# REGION

## Quick Reference

Converts an object that encloses an area into a region object

Draw 

Draw ➤ RegionAt the Command prompt, enter region.

**region**

Select objects: *Use an object selection method and press* ENTER *when you finish*

Regions are two-dimensional areas you create from closed shapes or loops. Closed polylines, lines, and curves are valid selections. Curves include circular arcs, circles, elliptical arcs, ellipses, and splines.

Closed 2D and exploded planar 3D polylines in the selection set are converted to separate regions and then converts polylines, lines, and curves to form closed planar loops (outer boundaries and holes of a region). If more than two curves share an endpoint, the resulting region might be arbitrary.

The boundary of the region consists of end-connected curves where each point shares only two edges. All intersections and self-intersecting curves are rejected.

If a selected polyline has been smoothed by either the Spline or Fit option of *PEDIT*, the resulting region contains the line or arc geometry of the smoothed polyline. The polyline is not converted to a spline object.

REGION deletes the original objects after converting them to regions unless the system variable *DELOBJ* is set to 0. If the original objects were hatched, hatch associativity is lost. To restore associativity, rehatch the region.

# REINIT

## Quick Reference

Reinitializes the digitizer, digitizer input/output port, and program parameters file

**reinit**

The Re-initialization dialog box (page 1182) is displayed.

## Re-initialization Dialog Box

### Quick Reference

**reinit**

Defines the I/O port and the device file for reinitialization.


**I/O Port Initialization** Reinitializes the I/O port for the digitizer.

**Device File Initialization** Reinitializes the digitizer and the *acad.pgp* file.

# RENAME

### Quick Reference

Changes the names of named objects

Format ➤ RenameAt the Command prompt, enter rename.
**rename**

The Rename dialog box (page 1182) is displayed.

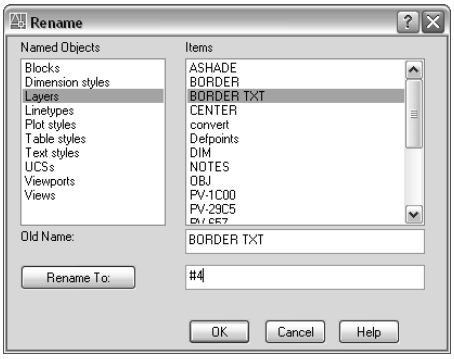
If you enter **-rename** at the command prompt, options are displayed at the command prompt (page 1183).

## Rename Dialog Box

### Quick Reference

Format ➤ RenameAt the Command prompt, enter rename.
**rename**

Renames named objects.




**Named Objects** Lists named objects in a drawing by category. Select the objects you want to rename.

**Items** Displays named objects of the type specified in Named Objects. Select the name you want to change.

**Old Name** Specifies the named object to be renamed. Enter a name or select a name from the Items list. You can use the wild-card characters * and ? to rename groups of objects. You can't rename some standard objects, such as layer 0 and the CONTINUOUS linetype

**Rename To** Specifies the new name you want to assign to the object. Enter a name and choose Rename To to apply the name change.

# -RENAME

## Quick Reference

If you enter **-rename** at the command prompt, the following RENAME command prompts are displayed.

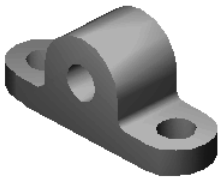
Enter object type to rename
[Block/Dimstyle/LAyer/LType/Material/Plotstyle/textStyle/Tablestyle/Ucs/VIew/VPort]:
*Enter a named object type to rename an object*
Enter old object name: *Enter the old name*
Enter new object name: *Enter the new name*

# RENDER

## Quick Reference

Creates a photorealistic or realistically shaded image of a three-dimensional wireframe or solid model


Render
View ➤ Render ➤ RenderAt the Command prompt, enter render.
**render**
Render panel, Render



The RENDER command begins the rendering process and displays the rendered image in the  (page 1184) or the viewport.

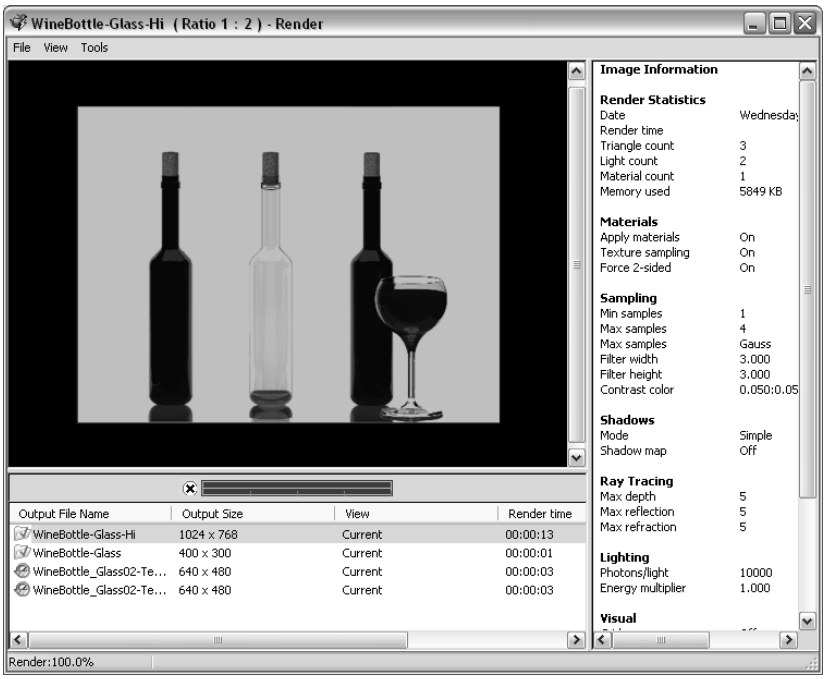
If you enter **-render** at the command prompt, options are displayed at the command prompt (page 1196).

# Render Window

## Quick Reference

**renderwin**
Render panel (click icon to expand), Show Render Window

The Render Window displays rendered output of the current model.




The Render Window is divided into the following three panes:

- **Image pane**. Displays the rendered image.
- **Statistics pane**. On the right side; shows the current settings used for the rendering.
- **History pane**. At the bottom; provides a recent history of images rendered from the current model and a progress meter to display rendering progress.

Separate render windows open for each drawing from which you are creating renderings. Rendering from any drawing always appears in its corresponding render window.

From the Render Window, you can

- Save the image to a file.
- Save a copy of the image to a file.
- Review the settings used for the current rendering.
- Track the rendering history of the model.

- Purge, delete, or purge and delete an image from the rendering history.
- Zoom into a section of the rendered image, pan around the image, and then zoom back out.

### File Menu

Saves rendered images.

**Save** Saves an image to a bitmap file. For more information, see "Save and Redisplay Rendered Images" in the *User's Guide*.

You cannot use the *SAVEIMG* command when rendering to the Render Window. This command applies only when you are rendering to a viewport.

**Save Copy** Saves a copy of an image to a new location without affecting the location stored with the current entry.

**Exit** Closes the Render Window.

If you re-open the render window, the history of past renderings is retained.

### View Menu

Displays different elements that make up the Render Window.

**Status Bar** Displays the status bar below the History pane.

**Statistics Pane** Displays the entire Statistics pane.

### Tools Menu

Provides commands for zooming into and out of a rendered image.

**Zoom +** Zooms into the rendering in the Image pane. When zoomed in, you can pan around the image.

**Zoom -** Zooms out of the rendering in the Image pane.

**TIP** If your mouse has a scroll wheel, you can zoom or pan the image by scrolling.

### Image Pane

The upper left portion of the Render Window is dedicated to the Image pane. It is the primary output target of the renderer. During rendering, the Image pane progressively displays the rendered tiles as they become available from the renderer.

Once rendering has completed, the Image pane provides additional functionality, such as

- **Scroll bars**. If the image does not fit into the space allocated to the Image pane, horizontal and vertical scrollbars appear at the bottom and right edges. Clicking the middle mouse button or wheel within the Image pane displays a hand icon. You can pan by dragging the cursor.
- **Zoom factor**. Initially, a small zoom factor is chosen so the image can fit in the Image pane. The zoom factor ranges from 1:64 to 64:1 in powers of two. You can change the zoom factor with the mouse wheel. Rolling the mouse forward increases zoom factor. The zoom factor is displayed in the title banner of the Render Window.

### Progress Meter / Status Bar

There are four phases that occur before an image is complete; translation, photon emission, final gather, and render. Between the Image pane and the History pane is a progress meter to give you an idea of how much each phase has been processed and how much of the overall image has been rendered. The progress meter is split to report two pieces of information.

- The top bar reports the progress within the current phase.
- The bottom bar shows the progress made toward completion of the entire rendering.

The translation and render phases always occur. Photon emission and final gather phases only occur if those options have been enabled. Processing time is only spent for those phases that are active.

You can also cancel a rendering by clicking the X icon next to the progress meter, or by pressing the ESC key.

A status bar, at the lower lift corner of the Render Window, echoes the progress of the top bar on the progress meter.

### History Pane

The lower left portion of the Render Window is dedicated to the History pane where you can browse a recent history of images rendered from the current

model. Each entry stored in the History pane is called a history entry. The data stored in history entries includes

- File name of the rendering and an indicator to tell you the type of rendering.
- Image size.
- View name that is being rendered. If no named view is used, the view is stored as *current view*.
- Render time measured in hours : minutes : seconds.
- Name of the render preset used for the rendering.

Any history entry that has a file name will be saved.

### History Pane - Shortcut Menu

Right-clicking on a history entry displays a menu that contains the following options:

**Render Again** Restarts the renderer for the selected history entry.

**Save** Displays the Render Output File dialog box (page 1190) where you can save the image to disk. If you've selected a temporary entry, saving converts it to a normal entry.

**Save Copy** Saves the image to a new location without affecting the location stored in the entry. The Render Output File dialog box (page 1190) is displayed.

**Make Render Settings Current** Loads the render settings associated with the selected history entry if multiple history entries are present that use different render presets.

**Remove From The List** Removes the entry from the history while leaving any associated image files in place.

**Delete Output File** Removes the rendered image from the Image pane.

### Statistics Pane

Provides an area, the right-hand portion of the Render Window, where you can inspect details about the rendering and render settings in effect when the image was created.

This information is derived from settings made in the Render Presets Manager dialog box (page 1203) along with information that is generated at the time of the rendering.

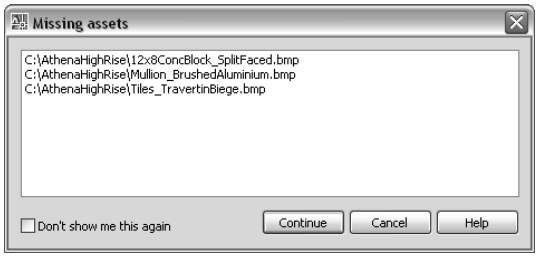
# Missing Texture Maps Dialog Box

## Quick Reference

Render
View ➤ Render ➤ RenderAt the Command prompt, enter render.
**render**
Render panel, Render

This dialog appears when you attempt to render a scene with texture maps whose path is no longer current. This can happen if the texture maps have been moved or deleted, or if the model has been placed on a system with a different drive mapping than the system on which it was created.

AutoCAD looks for texture maps in the folder where the drawing file is stored or in Texture Map Paths set on the Options dialog box.




**List of texture maps** Lists the texture maps that cannot be located, along with their path names.

**Don't show me this again** Check this option to not display this message the next time texture maps cannot be found.

**Continue** Renders the model anyway, without loading the missing texture maps.

# Render Output File Dialog Box

## Quick Reference

Render
View ➤ Render ➤ Advanced Render SettingsDoes not exist in the menus.
**rpref**
Render panel (click icon to expand), Output Size

The Render Output File dialog box is a standard file selection dialog boxes (page 931). You specify the file name of the image you want to save and the output file format. In this dialog box, you can only choose from raster image output file formats.

The file formats for saving your rendered images to include the following:

**BMP (*.bmp)** BMP files are still-image bitmap files in the Windows bitmap (*.bmp*) format.

Clicking Save after choosing this format displays the BMP Image Options dialog box (page 1191).

**PCX (*.pcx)** PCX files are relatively simple files that provide minimum compression using run length encoding (RLE).

Clicking Save after choosing this format displays the PCX Image Options dialog box (page 1192).

**TGA (*.tga)** The Targa (TGA) format supports up to 32-bit true color. This format is typically used as a true color format to render still images.

Clicking Save after choosing this format displays the Targa Image Options dialog box (page 1192).

**TIF (*.tif)** TIF (Tagged Image File) format is a multiplatform bitmap format. TIF is a common choice if you plan to send your output to a print service bureau or import the image into a page-layout program.

Clicking Save after choosing this format displays the TIFF Image Options dialog box (page 1193).

**JPEG (*.jpg)** JPEG (.jpeg or .jpg) files follow the standards set by the Joint Photography Experts Group. These files use a variable compression method that is called lossy compression because of the loss of image quality as you increase the compression. However, the JPEG compression scheme is extremely good and you can sometimes compress the file up to 200:1 without severe loss of image quality. JPEG is consequently a popular format for posting image files on the Internet for minimum file size and minimum download time.

Clicking Save after choosing this format displays the JPEG Image Options dialog box (page 1194).

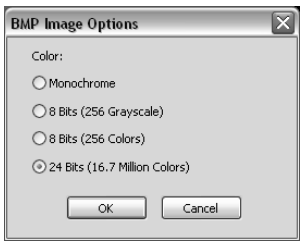
**PNG (*.png)** PNG (Portable Network Graphics) is a still-image file format developed for use with the Internet and World Wide Web. PNG is a format that generates a compressed image without any loss of quality, unlike the loss of quality found in JPEG files.

Clicking Save after choosing this format displays the PNG Image Options dialog box (page 1195).

# BMP Image Options Dialog Box

## Quick Reference

When BMP is chosen as the output format, the BMP Image Options dialog box is displayed.




### Color

**Monochrome** Creates a 2-bit black and white image.

**8 Bits (256 Grayscale)** Creates an 8-bit grayscale image using 256 shades of gray.

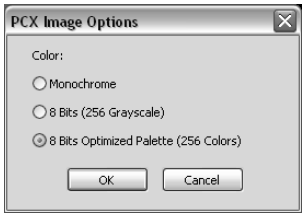
**8 Bits (256 Colors)** Renders a smaller, 8-bit color image from a palette of 256 colors.

**24 Bits (16.7 Million Colors)** Renders a larger, true color (24-bit) file.

# PCX Image Options Dialog Box

## Quick Reference

When PCX is chosen as the output format, the PCX Image Options dialog box is displayed.




### Color

**Monochrome** Creates a 2-bit black and white image.

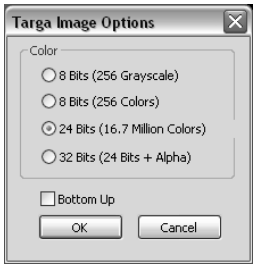
**8 Bits (256 Grayscale)** Creates an 8-bit grayscale image using 256 shades of gray.

**8 Bits (256 Colors)** Renders a smaller, 8-bit color image from a palette of 256 colors.

# Targa Image Options Dialog Box

## Quick Reference

When TGA is chosen as the output format, the Targa Image Options dialog box is displayed.



### Color

**8 Bits (256 Grayscale)** Creates an 8-bit grayscale image using 256 shades of gray.

**8 Bits (256 Colors)** Renders a smaller, 8-bit color image from a palette of 256 colors.

**24 Bits (16.7 Million Colors)** Creates a 24 bit color image that uses a 16.7 million color palette.

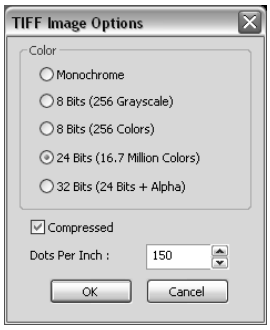
**32 Bits (24 Bits + Alpha)** Creates a 24 bit color image that includes an 8 bit alpha channel. Alpha is a type of data, found in 32-bit image files, that assigns transparency to the pixels in the image.

**Bottom Up** Saves the image from bottom to top.

## TIFF Image Options Dialog Box

### Quick Reference

When TIF is chosen as the output format, the TIFF Image Options dialog box is displayed.




### Color

**Monochrome** Creates a 2-bit black and white image.

**8 Bits (256 Grayscale)** Creates an 8-bit grayscale image using 256 shades of gray.

**8 Bits (256 Colors)** Renders a smaller, 8-bit color image from a palette of 256 colors.

**24 Bits (16.7 Million Colors)** Creates a 24 bit color image that uses a 16.7 million color palette.

**32 Bits (24 Bits + Alpha)** Creates a 24 bit color image that includes an 8 bit alpha channel. Alpha is a type of data, found in 32-bit image files, that assigns transparency to the pixels in the image.

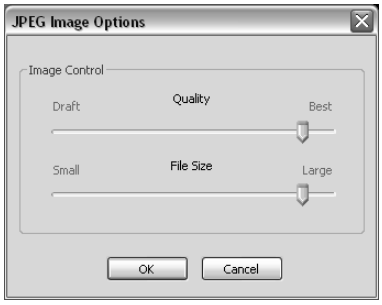
**Compressed** Applies lossless compression to the file.

**Dots Per Inch** Sets the dots per inch (dpi) for the saved image. This setting does not change the resolution of the final image, but can affect the way it prints in documents.

# JPEG Image Options Dialog Box

## Quick Reference

When JPG is chosen as the output format, the JPEG Image Options dialog box is displayed.




### Image Control

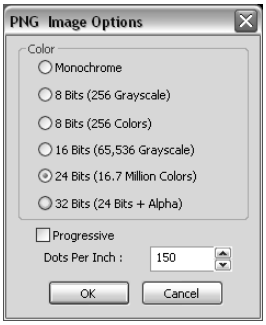
**Quality** Sets the level of quality: the higher the quality, the larger the file size. In general, files compressed with the slider set to Best have compression ratios between 5:1 and 15:1.

**File Size** Sets the size of file you want: the larger the file, the higher the quality.

# PNG Image Options Dialog Box

## Quick Reference

When PNG is chosen as the output format, the PNG Image Options dialog box is displayed.




### Color

**Monochrome** Creates a 2-bit black and white image.

**8 Bits (256 Grayscale)** Creates an 8-bit grayscale image using 256 shades of gray.

**8 Bits (256 Colors)** Renders a smaller, 8-bit color image from a palette of 256 colors.

**16 Bits (65,536 Grayscale)** Creates a grayscale 16-bit image that uses 65,536 shades of gray.

**24 Bits (16.7 Million Colors)** Creates a larger, true color (24-bit) file.

**32 Bits (24 Bits + Alpha)** Creates a 24 bit color image that includes an 8 bit alpha channel. Alpha is a type of data, found in 32-bit image files, that assigns transparency to the pixels in the image.

**Progressive** Provides faster display in Web browsers.

**Dots Per Inch** Sets the dots per inch (dpi) for the saved image. This setting does not change the resolution of the final image, but can affect the way it prints in documents.

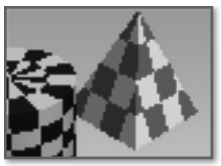
# -RENDER

## Quick Reference

If you enter **-render** at the command prompt, the following RENDER command prompts are displayed.

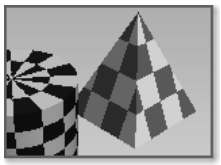
Specify render preset [Draft (page 1196)/Low (page 1196)/Medium (page 1197)/High (page 1197)/Presentation (page 1197)/Other (page 1197)] <Medium>: *Enter an option or press* ENTER
Specify render destination [Render Window (page 1198)/Viewport (page 1198)] <Render Window>: *Enter an option or press* ENTER

### Draft


Draft is the lowest level standard render preset. This setting is intended for very fast, test renderings where anti-aliasing is bypassed and sample filtering is very low.

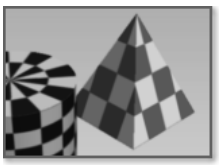
This render preset produces very low quality rendering, but results in the fastest rendering speed.

### Low


The Low render preset provides better quality than the Draft preset. Anti-aliasing is bypassed but sample filtering is improved. Raytracing is also active, by default, so better quality shading occurs.

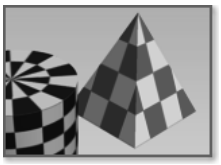
This preset is best used for test rendering that requires better quality than Draft.

### Medium


You can expect much better sample filtering and anti-aliasing is active when you use the Medium render preset. Raytracing is active with increased reflection depth settings when compared to the Low render preset.

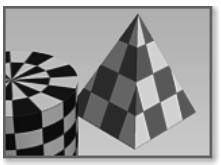
This preset offers a good balance between quality and render speed.

### High


The High render preset matches the Medium preset settings with regards to anti-aliasing, but sample filtering and raytracing is improved.

Due to the improved sample filtering and raytracing, rendered images take longer to process, but the image quality is much better.

### Presentation


The Presentation render preset is used for high quality, photo-realistic rendered images and takes the longest to process. Sample filtering and raytracing is further improved.

Since this preset is used for final renderings, global illumination settings are commonly used in conjunction.

### Other

The Other option allows you to specify a custom render preset if one or more are present. When you choose Other, the following prompt is displayed.

Specify custom render preset [? (page 1198)]: *Enter the name of a custom render preset or enter* ?

#### ?—List Custom Render Presets

A text screen is displayed listing all the custom render presets that are stored with the model. Only custom render presets are listed.

If no custom render presets are present, the text screen displays a message stating that no custom render preset are found.

Pressing Enter without specifying a custom render preset takes you back to the first prompt asking you to specify a render preset.

#### Render Window

Choosing Render Window as your render destination means the image will be displayed in the render window when processing in complete. Additional prompts appear when you choose Render Window.

Enter output width <640>: *Enter the desired output width or press* ENTER
Enter output height <480>: *Enter the desired output height or press* ENTER

The output width and height values designate how wide and how tall the rendered image is going to be. Both values are measured in pixels.

Save rendering to a file? [Yes/No] <No>: *Enter Y if you want the rendered image saved to disk or press* ENTER

If you accept the default value of No, the Render Window is displayed and the image is rendered. Answering Yes results in another prompt:

Specify output file name and path: *Enter a valid file name and path where the rendered image is to be saved*

#### Viewport

If you choose Viewport, anything that is currently displayed in the viewport gets rendered.

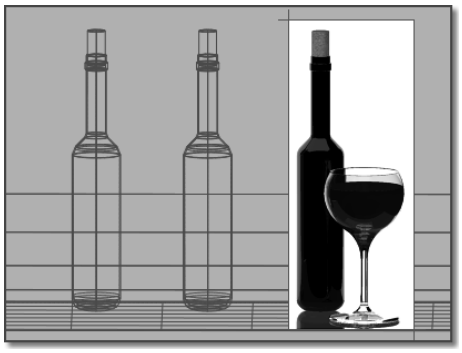
# RENDERCROP

## Quick Reference

Selects a specific region (crop window) in an image for rendering

**rendercrop**
Render panel, Render Cropped Region
Pick crop window to render: *Select the first corner of the region you want to render*
Please enter the second point: *Select the opposite corner of the region you want to render*

Renders what is inside a specified rectangular region within the viewport, and leaves the remainder of the render window intact. Use this command when you need to test render a part of the model.



The current render destination and rendering procedure is ignored. After the second corner of the render region is selected, the rendering task proceeds and displays a viewport rendering with cropping.

# RENDERENVIRONMENT

## Quick Reference

Provides visual cues for the apparent distance of objects

Render Toolbar ➤ 
View ➤ Render ➤ Render EnvironmentDoes not exist in the menus.
**renderenvironment**
Render panel (click icon to expand), Render Environment

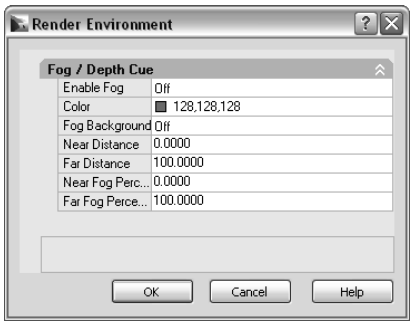
The Render Environment dialog box (page 1200) is displayed.

# Render Environment Dialog Box

## Quick Reference

Render Toolbar ➤
View ➤ Render ➤ Render EnvironmentDoes not exist in the menus.
**renderenvironment**

Defines the cues for distance between objects and the current viewing direction.




### Fog and Depth Cue

Fog and depth cueing are actually two extremes of the same effect: a white color is fog, and a black color is traditional depth cueing. You can use any color in between.

**Enable Fog** Turns fog on and off without affecting the other settings in the dialog box.

**Color** Specifies the color of the fog.

Clicking Select Color opens the Select Color dialog box (page 251). To define the color, you can select from the 255 AutoCAD Color Index (ACI) colors, true colors, and color book colors.

**Fog Background** Applies fog to the background as well as to the geometry.

**Near Distance** Specifies the distance from the camera where the fog begins.

It is specified as a percentage of the distance to the far clipping plane. You can set the value either by entering the Near Distance field or by using the

spinner. The Near Distance setting cannot be greater than the Far Distance setting.

**Far Distance** Specifies the distance from the camera where the fog ends.

It is specified as a percentage of the distance to the far clipping plane. You can set the value either by entering the Near Distance field or by using the spinner. The Far Distance setting cannot be less than the Near Distance setting.

**Near Fog Percentage** Specifies the opacity of the fog at the near distance.

**Far Fog Percentage** Specifies the opacity of the fog at the far distance.

# RENDEREXPOSURE

## Quick Reference

Provides settings to interactively adjust the global lighting for the most recent rendered output

**renderexposure**

The Adjust Render Exposure dialog box (page 1201) is displayed.

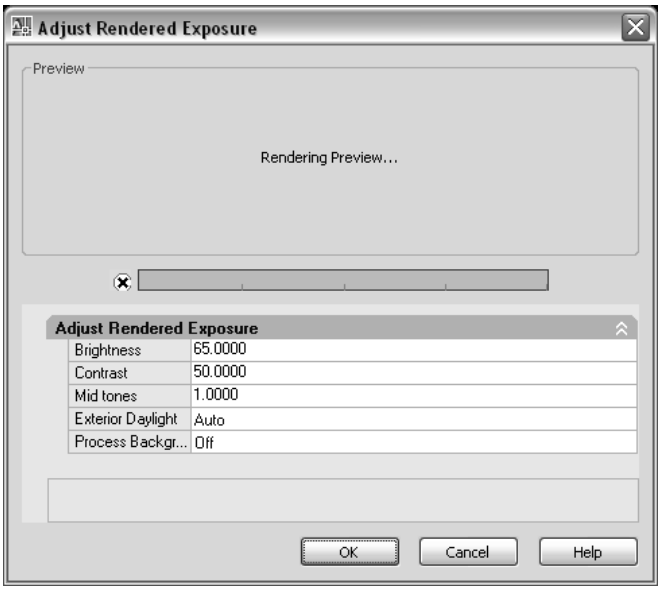
**NOTE** When the LIGHTINGUNITS system variable is set to 0, the Adjust Render Exposure dialog box is not available because the exposure control is not enabled.

# Adjust Render Exposure Dialog Box

## Quick Reference

**renderexposure**

Globally defines the brightness, contrast, mid tones, and exterior daylight for the current drawing when rendering.




### Adjust Render Exposure

Adjusting the exposure of the most recent rendered output allows you to add or remove lighting globally without the need to render the drawing over again each time you make a change to see how it affects the rendered output.

**Brightness** Adjusts the brightness of the converted colors. Values are 0-200.0. Default is [65.0].

**Contrast** Adjusts the brightness of the converted colors. Values are 0-100.0. Default is[100.0].

**Mid tones** Adjusts the mid-tone values of the converted colors. Values are 0-20.0. Default is [1.0].

**Exterior Daylight** Sets the exposure for exterior scenes lit by the sun. Values are On|Off|Auto. Default is [Auto].

**Process Background** Specifies if the background should be processed by exposure control at render time. Values are On|Off. Default is [On].

# RENDERPRESETS

## Quick Reference

Specifies render presets, reusable rendering parameters, for rendering an image

Render
**renderpresets**
Render panel, Select Render Preset ➤ Manage Render Presets

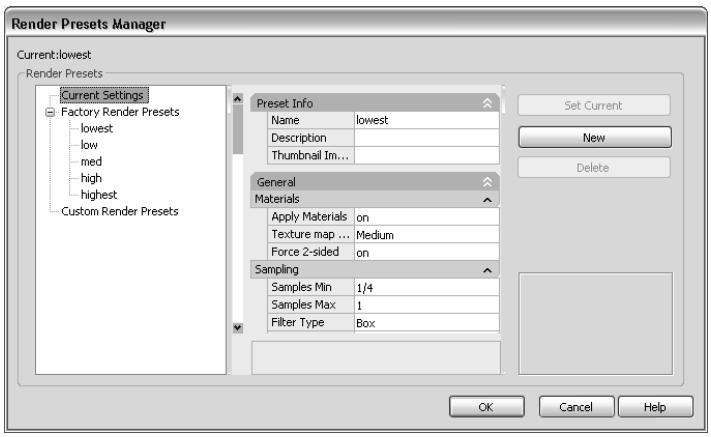
The Render Presets Manager (page 1203) is displayed.

# Render Presets Manager

## Quick Reference

Render
**renderpresets**
Render panel, Select Render Preset ➤ Manage Render Presets

Reusable rendering parameters are stored as *render presets*. You can choose from among a set of installed render presets or you can create your own custom render presets. Render presets are normally tailored for relatively quick, preview renderings. Others might be created for slower but higher quality renderings.




The Render Presets Manager is divided into four sections: a preset list, a property panel, button controls, and a thumbnail viewer.

### Render Presets List

Displays a tree view listing of all the presets stored with the current drawing. There are two types of render presets: Standard and Custom.

The Standard and Custom Render Presets trees can be re-ordered by dragging. Likewise, if you have multiple custom presets, you can re-order them in the same way. You cannot re-order the Standard presets inside the Standard Render Presets list.

### Property Panel

Provides settings that are similar to the properties on the Advanced Render Settings palette (page 1226).

### Preset Info

Provides general information about a selected preset.

**Name** Specifies the name of the selected render preset. You can also rename a custom preset. Standard presets cannot be renamed.

**Description** Displays a description of the selected preset.

**Thumbnail Image** Lists the name of a still image that is associated with the select preset. This gives you a visual idea of how the render settings will affect the rendered image.

Click [...] to open the Specify An Image dialog box (a standard file selection dialog box (page 931)) where you can browse to and select a thumbnail image for presets you create.

### Materials

Contains settings that affect how materials are handled by the renderer.

**Apply Materials** Applies the surface materials that you define and apply to an object. If Apply Materials is not selected, all objects in the drawing assume the color, ambient, diffuse, reflection, roughness, transparency, refraction, and bump map attribute values defined for the GLOBAL material. For more information, see *MATERIALS*.

**Texture Filtering** Specifies if texture maps are filtered.

**Force 2-Sided** Controls if both sides of faces are rendered.

### Sampling

Controls how the renderer performs sampling.

**Min Samples** Sets the minimum sample rate. The value represents the number of samples per pixel. A value greater than or equal to 1 indicates that one or more samples are computed per pixel. A fractional value indicates that one sample is computed for every N pixels (for example, 1/4 computes a minimum of one sample for every four pixels). Default=1/4.

**Max Samples** Sets the maximum sample rate. If neighboring samples find a difference in contrast that exceeds the contrast limit, the area containing the contrast is subdivided to the depth specified. Default=1.

The values of the Min Samples and Max Samples lists are "locked" together so that the value of Min Samples can't exceed the value of Max Samples. An error dialog box is displayed if the Min Samples value is greater than the Max Samples value.

**Filter Type** Determines how multiple samples are combined into a single pixel value. The filter types are:

- **Box**. Sums all samples in the filter area with equal weight. This is the quickest sampling method.
- **Gauss**. Weights the samples using a Gauss (bell) curve centered on the pixel.
- **Triangle**. Weights the samples using a pyramid centered on the pixel.
- **Mitchell**. Weights the samples using a curve (steeper than Gauss) centered on the pixel.
- **Lanczos**. Weights the samples using a curve (steeper than Gauss) centered on the pixel, diminishing the effect of samples at the edge of the filter area.

**Filter Width and Height** Specifies the size of the filtered area. Increasing the value of Filter Width and Filter Height can soften the image; however, it will increase rendering time.

**Contrast Color** Clicking [...] opens the Select Color dialog box (page 251) where you interactively specify the R,G,B threshold values.

**Contrast Red, Blue, Green** Specifies the threshold values for the red, blue, and green components of samples. These values are normalized, and range from 0.0 to 1.0, where 0.0 indicates the color component is fully unsaturated

(black, or 0 in eight-bit encoding) and 1.0 indicates the color component is fully saturated (white, or 255 in eight-bit encoding).

**Contrast Alpha** Specifies the threshold value for the alpha component of samples. This value is normalized, and ranges from 0.0 (fully transparent, or 0 in eight-bit encoding) to 1.0 (fully opaque, or 255 in eight-bit encoding).

### Shadows

Contains settings that affect how shadows appear in the rendered image.

**Enable** Specifies if shadows are computed during rendering.

**Mode** The shadow mode can be Simple, Sort, or Segments.

- **Simple**. Generates shadow shaders in a random order.
- **Sort**. Generates shadow shaders in order, from the object to the light.
- **Segments**. Generates shadow shaders in order along the light ray from the volume shaders to the segments of the light ray between the object and the light.

**Shadow Map** Controls if shadow mapping is used to render shadows. When on, the renderer renders shadow-mapped shadows. When off, all shadows are ray-traced.

### Ray Tracing

Contains settings that affect the shading of a rendered image.

**Enable** Specifies if ray tracing should be performed when shading.

**Max Depth** Limits the combination of reflection and refraction. Tracing of a ray stops when the total number of reflections and refractions reaches the maximum depth. For example, if Max Depth equals 3 and the two trace depths each equal the default value of 2, a ray can be reflected twice and refracted once, or vice versa, but it cannot be reflected and refracted four times.

**Max Reflections** Sets the number of times a ray can be reflected. At 0, no reflection occurs. At 1, the ray can be reflected once only. At 2, the ray can be reflected twice, and so on.

**Max Refractions** Sets the number of times a ray can be refracted. At 0, no refraction occurs. At 1, the ray can be refracted once only. At 2, the ray can be refracted twice, and so on.

### Global Illumination

Affects how your scene is illuminated.

**Enable** Specifies if global illumination should be calculated for the scene.

**Photons/Samples** Sets how many photons are used to compute the intensity of the global illumination. Increasing this value makes global illumination less noisy but also more blurry. Decreasing this value makes global illumination more noisy but less blurry. The larger the Samples value, the greater the rendering time.

**Use Radius** Determines the size of photons. When on, the spinner value sets the size of photons. When off, each photon is calculated to be 1/10 of the radius of the full scene.

**Radius** Specifies the area within which photons will be used when illuminance is computed.

**Max Depth** Limits the combination of reflection and refraction. Reflection and refraction of a photon stop when the total number of both equals the Max Depth setting. For example, if Max Depth equals 3 and the trace depths each equal 2, a photon can be reflected twice and refracted once, or vice versa, but it can't be reflected and refracted four times.

**Max Reflections** Sets the number of times a photon can be reflected. At 0, no reflection occurs. At 1, the photon can be reflected once only. At 2, the photon can be reflected twice, and so on.

**Max Refractions** Sets the number of times a photon can be refracted. At 0, no refraction occurs. At 1, the photon can be refracted once only. At 2, the photon can be refracted twice, and so on.

### Final Gather

Calculates global illumination further.

**Enable** Specifies if gathering should be used to compute the final shading.

**Rays** Sets how many rays are used to compute indirect illumination in a final gather. Increasing this value makes global illumination less noisy, but also increases rendering time.

**Radius Mode** Determines the radius mode for final gather processing. Settings are On, Off, or View.

- **On**. Specifies that the setting means the Max Radius setting is used for final gather processing. The radius is specified in world units, and defaults to 10 percent of the maximum circumference of the model.
- **Off**. Specifies the maximum radius is the default value of 10 percent of the maximum model radius, in world units.
- **View**. Specifies the Max Radius setting in pixels instead of world units and is used for final gather processing.

**Max Radius** Sets the maximum radius within which final gathering is used. Reducing this value can improve quality at a cost of increased rendering time.

**Use Min** Controls whether the Min Radius setting is used during final gather processing. When on, the minimum radius setting is used for final gather processing. When off, the minimum radius is not used.

**Min Radius** Sets the minimum radius within which final gathering is used. Increasing this value can improve quality but increase rendering time.

### Light Properties

Affects how lights behave when calculating indirect illumination. By default, the energy and photon settings apply to all lights in a scene.

**Photons/Light** Sets the number of photons emitted by each light for use in global illumination. Increasing this value increases the accuracy of global illumination, but also increases the amount of memory used and the length of render time. Decreasing this value improves memory usage and render time, and can be useful for previewing global-illumination effects.

**Energy Multiplier** Multiplies the global illumination, indirect light, intensity of the rendered image.

### Visual

Helps you understand why the renderer is behaving in a certain way.

**Grid** Renders an image that shows the coordinate space of objects, the world, or camera.

- **Object**. Shows local coordinates (UVW). Each object has its own coordinate space.

- **World**. Shows world coordinates (XYZ). The same coordinate system applies to all objects.
- **Camera**. Shows camera coordinates, which appear as a rectangular grid superimposed on the view.

**Grid Size** Sets the size of the grid.

**Photon** Renders the effect of a photon map. This requires that a photon map be present. If no photon map is present, the Photon rendering looks just like the nondiagnostic rendering of the scene: the renderer first renders the shaded scene, then replaces it with the pseudocolor image.

- **Density**. Renders the photon map as it is projected into the scene. High density is displayed in red, and lower values render in increasingly cooler colors.
- **Irradiance**. Similar to the Density rendering, but shades the photons based on their irradiance. The maximum irradiance is rendered in red, and lower values render in increasingly cooler colors.

**BSP** Renders a visualization of the parameters used by the tree in the BSP raytrace acceleration method. If a message from the renderer reports excessively large depth or size values, or if rendering seems unusually slow, this can help you locate the problem.

- **Depth**. Shows the depth of the tree, with top faces in bright red, and increasingly deep faces in increasingly cool colors.
- **Size**. Shows the size of leaves in the tree, with differently sized leaves indicated by different colors.

### Processing

**Tile Size** Determines the tile size for rendering. To render the scene, the image is subdivided into tiles. The smaller the tile size, the more image updates are generated during rendering. When the tile size is reduced, the number of image updates increases, meaning that a rendering take longer to complete. If the tile size is increased, fewer image updates occur and the rendering takes less time to complete.

**Tile Order** Specifies the method used (render order) for tiles as an image is rendered. You can choose a method based on how you prefer to see the image appear as it renders in the Render Window.

- **Hilbert**. Next tile to be rendered is based on the cost of switching to the next one.
- **Spiral**. Tiles are rendered beginning at the center of the image, and spiral outward.
- **Left to Right**. Tiles are rendered in columns, from bottom to top, left to right.
- **Right to Left**. Tiles are rendered in columns, from bottom to top, right to left.
- **Top to Bottom**. Tiles are rendered in rows, from right to left, top to bottom.
- **Bottom to Top**. Tiles are rendered in rows, from right to left, bottom to top.

**Memory Limit** Determines the memory limit for rendering. The renderer keeps a count of the memory it uses at render time. If the memory limit is reached, the geometry for some objects is discarded in order to allocate memory for other objects.

### Set Current

Sets the selected render preset as the preset to be used by the renderer.

### Create Copy

Copies a preset. The Copy Render Preset dialog box (page 1211) is displayed.

### Delete

Deletes the selected custom render preset. Standard presets cannot be deleted.

### Thumbnail Viewer

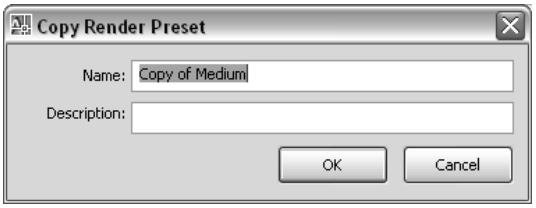
Displays a thumbnail image that is associated with the selected render preset. If the thumbnail image is not shown, you can select one from the Thumbnail Image setting under Preset Info.

## Copy Render Preset Dialog Box

### Quick Reference

Render
**renderpresets**
Render panel, Select Render Preset ➤ Manage Render Presets ➤ Create Copy

You can define your own custom presets by creating a copy based upon an existing render preset. You can specify a name and description of the copied preset. The new preset shows up as one of your Custom Render Presets in the Render Presets list in the Render Presets Manager.




**Name** Give the copied preset a name. Preset names must be unique.

**Description** Add a description of the custom preset.

Describe the rendering effect that will be produced when the custom preset is used. This description can be more detailed than the name of the preset.

---

**NOTE** Render preset names cannot include special characters. A warning dialog box is displayed if a special character is found in your preset name.

---

After you define your new render preset, you can select it in the Render Presets list and alter the rendering settings you want stored with that preset.

# RENDERWIN

### Quick Reference

Displays the Render Window without invoking a render task

Render panel (click icon to expand), Show Render Window
**renderwin**

The Render Window (page 1184) is displayed, but a rendering of the current drawing is not initiated. If the drawing contains a render history, you can view images you've previously rendered.

If the current drawing does not contain a rendering history, you are informed that no rendering history is present and the command ends.

# RENDSCR

## Quick Reference

Obsolete

**rendscr**

The Render Window (page 1184) is displayed. (*RENDERWIN* command)

# REPLAY

## Quick Reference

Obsolete

**replay**

This command is obsolete.

# RESETBLOCK

## Quick Reference

Resets one or more dynamic block references to the default values of the block definition

Select a dynamic block reference in a drawing. Right-click in the drawing area. Click Reset Block.
**resetblock**
Select object: *Select one or more dynamic block references and press ENTER*

# RESUME

## Quick Reference

Continues an interrupted script

**resume** (or '**resume** for transparent use)

You can interrupt a macro script that is running by pressing ESC or BACKSPACE. Any error encountered while processing input from a script file causes the script to be suspended. If a script is suspended while the program is active, you can use RESUME to continue the script.

# REVCLOUD

## Quick Reference

Creates a polyline of sequential arcs to form a cloud shape

Draw 

Draw ➤ Revision CloudAt the Command prompt, enter revcloud.

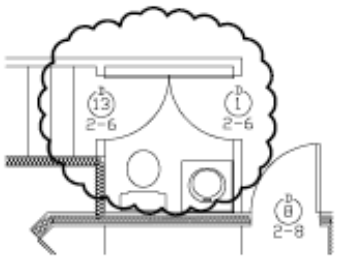
**revcloud**

Minimum arc length: 0.5000 Maximum arc length: 0.5000
Specify start point or [Arc length (page 1214)/Object (page 1214)/Style (page 1214)]
<Object>: *Drag to draw the revision cloud, enter an option, or press* ENTER
Guide crosshairs along cloud path...

When the start and end lines meet, the following message is displayed at the command prompt.

Revision cloud finished

The resulting object is a polyline.



**NOTE** REVCLOUD stores the last used arc length in the system registry. This value is multiplied by *DIMSCALE* to provide consistency when the program is used with drawings that have different scale factors.

### Arc Length

Specifies the length of the arcs in a revision cloud.

Specify minimum length of arc <0.5000>: *Specify a minimum arc length*
Specify maximum length of arc <0.5000>: *Specify a maximum arc length*
Guide crosshairs along cloud path...
Revision cloud finished

The maximum arc length cannot be set to more than three times the minimum arc length.

### Object

Specifies an object to be converted to a revision cloud.

Select object: *Select the closed object to convert to a revision cloud*
Reverse direction [Yes/No]: *Enter* **y** *to reverse the direction of the arcs in the revision cloud, or press* ENTER *to leave the arcs as is*
Revision cloud finished

### Style

Specifies the style of the revision cloud.

Select arc style [Normal/Calligraphy] <default/last>: *Select the style for the revision cloud*

# REVOLVE

## Quick Reference

Creates a 3D solid or surface by revolving 2D objects about an axis

Modeling 
Draw ➤ Modeling ➤ RevolveAt the Command prompt, enter revolve.
**revolve**
3D Make panel, Revolve
Current wire frame density: ISOLINES=4
Select objects to revolve: *Use an object selection method*

With the REVOLVE command, you can create a new solid or surface by revolving an open or closed planar curve about an axis. You can revolve more than one object.

The *DELOBJ* system variable controls whether revolved objects are automatically deleted when the solid or surface is created or whether you are prompted to delete the objects.

You can select the objects to revolve before you start the command.

You can revolve the following objects:

- Lines
- Arcs
- Elliptical arcs
- 2D polylines
- 2D splines
- Circles
- Ellipses
- Planar 3D faces
- 2D solids
- Traces
- Regions

■ Planar faces on solids or surfaces

**NOTE** You can select faces on solids by pressing and holding CTRL, and then selecting these subobjects.

You cannot revolve objects contained within a block. Polylines that have crossing or self-intersecting segments cannot be revolved. REVOLVE ignores the width of a polyline and revolves from the center of the path of the polyline.

The right-hand rule determines the positive direction of rotation. See Control the User Coordinate System in 3D in the *User's Guide*.

Specify axis start point (page 1216) or define axis by [Object (page 1217)/X (page 1217)/Y (page 1217)/Z (page 1218)] <Object>:*Specify a point, press* ENTER *to select an object for the axis, or enter an option*
**Axis Start Point** Specifies the first and second points of the axis of revolution. The positive axis direction is from the first to the second point.

Specify axis endpoint: *Specify a point (2)*

Specify angle of revolution or [STart angle] <360>: *Specify an angle or press* ENTER

A positive angle revolves the objects in a counterclockwise direction. A negative angle revolves the objects in a clockwise direction.

The objects are revolved to the specified angle.


selected axis points | full circle | specified angle

### Start Angle

Specifies an offset for the revolution from the plane of the object being revolved.

Specify start angle <0>: *Specify an angle or press* ENTER
Specify angle of revolution <360>: *Specify an angle or press* ENTER

**Object** Allows you to select an existing object that defines the axis about which to revolve the selected object. The positive axis direction is from the closest to the farthest endpoint of this object.

The following objects can be used as an axis:

- Lines
- Linear polyline segments
- Linear edges of solids or surfaces

---

**NOTE** You can select an edge on a solid by pressing and holding CTRL, and then selecting an edge.

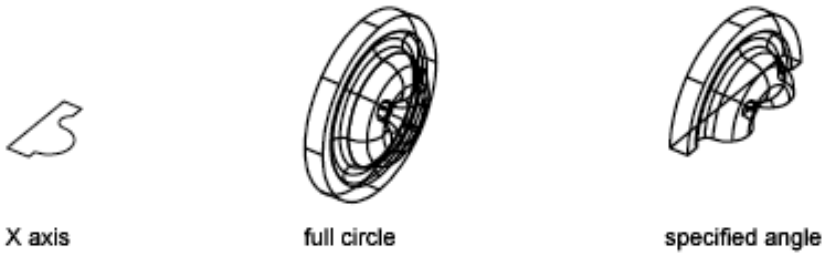
---

Select an object: *Use an object selection method*

Specify angle of revolution or [STart angle] <360>: *Specify an angle or press* ENTER


**X (Axis)** Uses the positive *X* axis of the current UCS as the positive axis direction.

Specify angle of revolution or [STart angle] <360>: *Specify an angle or press* ENTER




**Y (Axis)** Uses the positive *Y* axis of the current UCS as the positive axis direction.

Specify angle of revolution or [STart angle] <360>: *Specify an angle or press* ENTER

Y axis

full circle

specified angle

**Z (Axis)** Uses the positive *Z* axis of the current UCS as the positive axis direction.

Specify angle of revolution or [STart angle] <360>: *Specify an angle or press* ENTER

# REVSURF

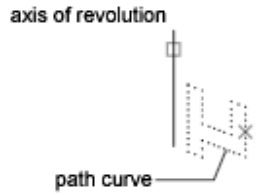
## Quick Reference

Creates a revolved mesh about a selected axis

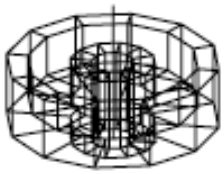
Draw ➤ Modeling ➤ Meshes ➤ Revolved MeshAt the Command prompt, enter revsurf.

**revsurf**

Current wire frame density: SURFTAB1=*current*: SURFTAB2=*current*
Select object to revolve: *Select a line, arc, circle, or 2D or 3D polyline*
Select object that defines axis of revolution: *Select a line or open 2D or 3D polyline*




The path curve is swept about the selected axis to define the mesh. The path curve defines the *N* direction of the mesh. Selecting a circle or a closed polyline as the path curve closes the mesh in the *N* direction.



The vector from a polyline's first vertex to its last vertex determines the rotation axis. Any intermediate vertices are ignored. The axis of revolution determines the *M* direction of the mesh.

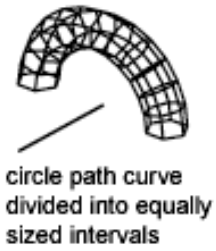
Specify start angle <0>: *Enter a value or press* ENTER
Specify included angle (+=ccw, -=cw) <360>: *Enter a value or press* ENTER
**Start Angle** If set to a nonzero value, begins the mesh of revolution at an offset from the generating path curve.

**Included Angle** Specifies how far about the axis of revolution the mesh extends.

Specifying a start angle begins the mesh of revolution at an offset from the generating path curve. The included angle is the distance through which the path curve is swept.

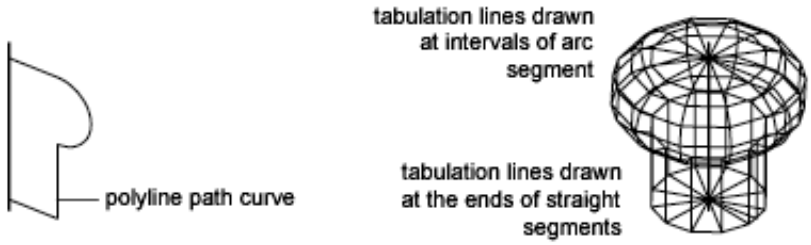
Entering an included angle that is less than a full circle prevents the circle from closing.


The point you use to select the axis of revolution affects the direction of revolution. Each of the meshes in the examples below was created by specifying a start angle of 0 degrees and an included angle of 90 degrees.




The density of the generated mesh is controlled by the *SURFTAB1* and *SURFTAB2* system variables. SURFTAB1 specifies the number of tabulation lines that are drawn in the direction of revolution. If the path curve is a line, arc, circle, or spline-fit polyline, SURFTAB2 specifies the number of tabulation lines that are drawn to divide it into equal-sized intervals. If the path curve is

a polyline that has not been spline fit, tabulation lines are drawn at the ends of straight segments, and each arc segment is divided into the number of intervals specified by SURFTAB2.




# RMAT

## Quick Reference

Obsolete

**rmat**

The Materials window (page 794) is displayed. (*MATERIALS* command)

# ROTATE

## Quick Reference

Revolves objects around a base point

Modify 
Modify ➤ RotateDoes not exist in the menus.
Select the objects to rotate, and right-click in the drawing area. Click Rotate.
**rotate**
Current positive angle in UCS: ANGDIR=*current* ANGBASE=*current*
Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify base point: *Specify a point*
Specify rotation angle (page 1221) or [Copy (page 1221)/Reference (page 1221)]: *Enter an angle, specify a point, enter* **c**, *or enter* **r**

**Rotation Angle** Determines how far an object rotates around the base point.

The axis of rotation passes through the specified base point and is parallel to the *Z* axis of the current UCS.

**Copy** Creates a copy of the selected objects for rotation.

**Reference** Rotates objects from a specified angle to a new, absolute angle.

Specify the reference angle <*last reference angle*>: *Specify an angle by entering a value or by specifying two points*

Specify the new angle or [Points] <*last new angle*>: *Specify the new absolute angle by entering a value or by specifying two points*

When you rotate a viewport object, the borders of the viewport remain parallel to the edges of the drawing area.

# ROTATE3D

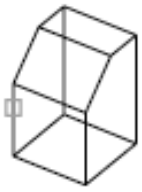
## Quick Reference

Moves objects about a three-dimensional axis

**rotate3d**

It is recommended that you use the grip tools available through the *3DMOVE* and *3DROTATE* commands to manipulate 3D objects. For more information about using grip tools, see Use Grip Tools to Modify Objects.

Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify first point on axis or define axis by [Object (page 1221)/Last (page 1223)/View (page 1223)/Xaxis/Yaxis/Zaxis (page 1224)/2points (page 1224)]: *Specify a point, enter an option, or press* ENTER



**Object**

Aligns the axis of rotation with an existing object.

Select a line, circle, arc or 2D-polyline segment:

### Line

Aligns the axis of rotation with the line selected.

**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### Circle

Aligns the axis of rotation with the 3D axis of the circle (perpendicular to the plane of the circle and passing through the center of the circle).

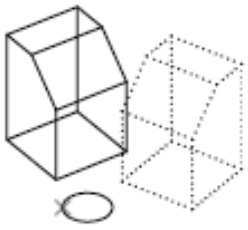
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.




### Arc

Aligns the axis of rotation with the 3D axis of the arc (perpendicular to the plane of the arc and passing through the center of the arc).

**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### 2D Polyline Segment

Aligns the axis of rotation with a segment of the polyline. Treats a straight segment as a line segment. Treats an arc segment as an arc.

Specify rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### Last

Uses the last axis of rotation.

Specify rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### View

Aligns the axis of rotation with the viewing direction of the current viewport that passes through the selected point.

Specify a point on the view direction axis <0,0,0>:

Specify rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### X Axis,Y Axis, Z Axis

Aligns the axis of rotation with one of the axes (*X*, *Y*, or *Z*) that pass through the selected point.

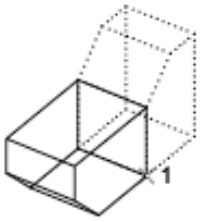
Specify a point on the (X, Y, or Z) axis <0,0,0>: *Specify a point (1)*
Specify rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

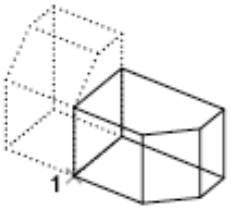
Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

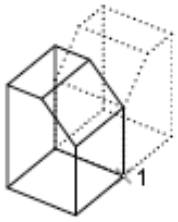
The difference between the starting angle and the ending angle is the computed rotation angle.



X axis



Y axis



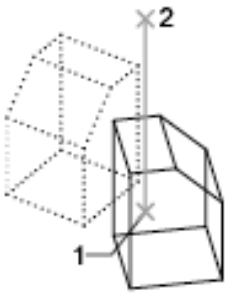
Z axis

### 2 Points

Uses two points to define the axis of rotation. Pressing ENTER at the main ROTATE3D prompt displays the following prompts. Specifying a point at the main prompt skips the prompt for the first point.

Specify first point on axis: *Specify a point (1)*
Specify second point on axis: *Specify a point (2)*

Specify rotation angle or [Reference]: *Specify an angle or enter* **r**




**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference angle <0>: *Specify the starting angle*

Specify the new angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

# RPREF

## Quick Reference

Displays the Advanced Render Settings palette for access to advanced rendering settings

Render
View ➤ Render ➤ Advanced Render SettingsDoes not exist in the menus.
Tools ➤ Palettes ➤ Advanced Render SettingsDoes not exist in the menus.
**rpref**
Render panel - expanded ➤ Advanced Render Settings

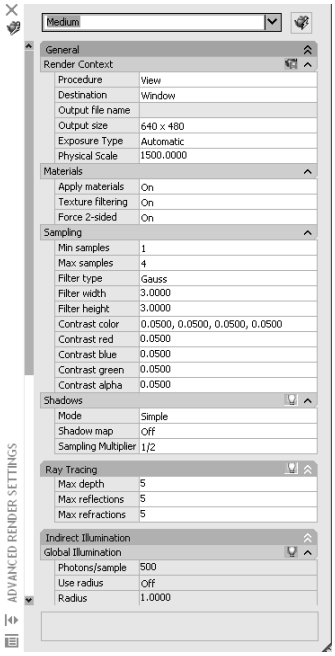
The Advanced Render Settings palette (page 1226) is displayed.

# Advanced Render Settings Palette

## Quick Reference

Render
View ➤ Render ➤ Advanced Render SettingsDoes not exist in the menus.
**rpref**
Render panel - expanded ➤ Advanced Render Settings

You use the Advanced Render Settings palette to make rendering settings. You can also access the Render Presets Manager (page 1203) from the Render Settings palette.




### Render Preset List / Select Render Preset

Lists standard render presets ranging from lowest to highest quality, up to four custom render presets and allows access to the Render Presets Manager.

### Render

Renders the model directly from the Advanced Render Settings palette.

### Render Context

Contains settings that affect how your model gets rendered.

**Save File** Determines if the rendered image is written to a file.

**Rendering Procedure** Controls the model content that gets processed during rendering. The render procedure has three settings: View, Crop, and Selected.

- **View**. Renders the current view without displaying the Render dialog box.
- **Crop**. Creates a render area at render time. When you click the Render button with Crop Window selected, you are prompted to specify an area in the drawing before rendering proceeds. This option is available only when Viewport is selected under Destination.
- **Selected**. Displays a prompt to select objects to render.

**Destination** Determines the output site that the renderer uses to display the rendered image.

- **Window**. Renders to the Render Window (page 1184).
- **Viewport**. Renders to a viewport.

**Output File Name** Specifies a file name and location where the rendered image will be stored. The File Type list shows the following formats:

- **BMP (*.bmp)**. Still-image bitmap file in the Windows bitmap (*.bmp*) format.
- **PCX (*.pcx)**. Simple format that provides a minimum of compression.
- **TGA (*.tga)**. File format that supports 32-bit true color; that is, 24-bit color plus an alpha channel, and is typically used as a true color format.
- **TIF (*.tif)**. Multiplatform bitmap format.
- **JPEG (*.jpg)**. Popular format for posting image files on the Internet for minimum file size and minimum download time.
- **PNG (*.png)**. Still-image file format developed for use with the Internet and World Wide Web.

**Output Size** Shows the current output resolution setting for the rendered image. Opening the Output Size list displays the following:

- Up to four custom size settings.

  **NOTE** Custom output sizes are not stored with the drawing and they are not retained between drawing sessions.

- Four of the most commonly used output resolutions.
- Access to the Output Size dialog box. (page 1234)

**Exposure Type** Controls the tone operator setting. This does not need to be stored in the named render preset. Rather it can be stored per drawing in the render context.

- **Automatic.** Indicates that the tone operator used should be chosen to match the current viewport tone operator strategy.
- **Logarithmic.** Indicates that the log exposure control should be used.

**Physical Scale** Specifies the physical scale. Default = 1500.

### Materials

Contains settings that affect how materials are handled by the renderer.

**Apply Materials** Applies the surface materials that you define and attach to an object in the drawing. If Apply Materials is not selected, all objects in the drawing assume the color, ambient, diffuse, reflection, roughness, transparency, refraction, and bump map attribute values defined for the GLOBAL material. For more information, see *MATERIALS*.

**Texture Filtering** Specifies how texture maps are filtered.

**Force 2-Sided** Controls if both sides of faces are rendered.

### Sampling

Controls how the renderer performs sampling.

**Min Samples** Sets the minimum sample rate. The value represents the number of samples per pixel. A value greater than or equal to 1 indicates that one or more samples are computed per pixel. A fractional value indicates that one sample is computed for every N pixels (for example, 1/4 computes a minimum of one sample for every four pixels). Default=1/4.

**Max Samples** Sets the maximum sample rate. If neighboring samples find a difference in contrast that exceeds the contrast limit, the area containing the contrast is subdivided to the depth specified by Maximum. Default=1.

The values of the Min Samples and Max Samples lists are "locked" together so that the value of Min Samples can't exceed the value of Max Samples. An error dialog box is displayed if the Min Samples value is greater than the Max Samples value.

**Filter Type** Determines how multiple samples are combined into a single pixel value. The filter types are:

- **Box**. Sums all samples in the filter area with equal weight. This is the quickest sampling method.
- **Gauss**. Weights the samples using a Gauss (bell) curve centered on the pixel.
- **Triangle**. Weights the samples using a pyramid centered on the pixel.
- **Mitchell**. Weights the samples using a curve (steeper than Gauss) centered on the pixel.
- **Lanczos**. Weights the samples using a curve (steeper than Gauss) centered on the pixel, diminishing the effect of samples at the edge of the filter area.

**Filter Width and Filter Height** Specifies the size of the filtered area. Increasing the value of Filter Width and Filter Height can soften the image; however, it will increase rendering time.

**Contrast Color** Clicking [...] opens the Select Color dialog box (page 251) where you interactively specify the R,G,B threshold values.

**Contrast Red, Blue, Green** Specifies the threshold values for the red, blue, and green components of samples. These values are normalized, and range from 0.0 to 1.0, where 0.0 indicates the color component is fully unsaturated (black, or 0 in eight-bit encoding) and 1.0 indicates the color component is fully saturated (white, or 255 in eight-bit encoding).

**Contrast Alpha** Specifies the threshold value for the alpha component of samples. This value is normalized, and ranges from 0.0 (fully transparent, or 0 in eight-bit encoding) to 1.0 (fully opaque, or 255 in eight-bit encoding).

### Shadows

Contains settings that affect how shadows appear in the rendered image.

**Enable** Specifies if shadows are computed during rendering.

**Mode** The shadow mode can be Simple, Sort, or Segments.

- **Simple**. Generates shadow shaders in a random order.
- **Sort**. Generates shadow shaders in order, from the object to the light.
- **Segments**. Generates shadow shaders in order along the light ray from the volume shaders to the segments of the light ray between the object and the light.

**Shadow Map** Controls if shadow mapping is used to render shadows. When on, the renderer renders shadow-mapped shadows. When off, all shadows are ray-traced.

**Sampling Multiplier** Globally limits shadow sampling for area lights. This is part of the rendering preset data. This allows draft and low quality presets to reduce area light sampling. It's effect is to modulate the inherent sampling frequency specified for each light. The default value=1 for new presets. Values are 0, 1/8, 1/4, 1/2, 1, 2. Draft: 0; Low:1/4; Med:1/2; High:1; Presentation:1.

### Ray Tracing

Contains settings that affect the shading of a rendered image.

**Enable** Specifies if ray tracing should be performed when shading.

**Max Depth** Limits the combination of reflection and refraction. Tracing of a ray stops when the total number of reflections and refractions reaches the maximum depth. For example, if Max Depth equals 3 and the two trace depths each equal the default value of 2, a ray can be reflected twice and refracted once, or vice versa, but it cannot be reflected and refracted four times.

**Max Reflections** Sets the number of times a ray can be reflected. At 0, no reflection occurs. At 1, the ray can be reflected once only. At 2, the ray can be reflected twice, and so on.

**Max Refractions** Sets the number of times a ray can be refracted. At 0, no refraction occurs. At 1, the ray can be refracted once only. At 2, the ray can be refracted twice, and so on.

### Global Illumination

Affects how your scene is illuminated.

**Enable** Specifies if lights should cast indirect light into the scene.

**Photons/Samples** Sets how many photons are used to compute the intensity of the global illumination. Increasing this value makes global illumination less noisy but also more blurry. Decreasing this value makes global illumination more noisy but less blurry. The larger the Samples value, the greater the rendering time.

**Use Radius** Determines the size of photons. When on, the spinner value sets the size of photons. When off, each photon is calculated to be 1/10 of the radius of the full scene.

**Radius** Specifies the area within which photons will be used when illuminance is computed.

**Max Depth** Limits the combination of reflection and refraction. Reflection and refraction of a photon stop when the total number of both equals the Max Depth setting. For example, if Max Depth equals 3 and the trace depths each equal 2, a photon can be reflected twice and refracted once, or vice versa, but it can't be reflected and refracted four times.

**Max Reflections** Sets the number of times a photon can be reflected. At 0, no reflection occurs. At 1, the photon can be reflected once only. At 2, the photon can be reflected twice, and so on.

**Max Refractions** Sets the number of times a photon can be refracted. At 0, no refraction occurs. At 1, the photon can be refracted once only. At 2, the photon can be refracted twice, and so on.

### Final Gather

Calculates global illumination.

**Mode** Controls the final gathering dynamic settings.

- **On**. Turns on the global illumination in final gather.
- **Off**. Turns off the calculation of global illumination in final gather.
- **Auto.** Indicates that the final gather should be dynamically enabled or disable at render time based on the skylight status.

**Rays** Sets how many rays are used to compute indirect illumination in a final gather. Increasing this value makes global illumination less noisy, but also increases rendering time.

**Radius Mode** Determines the radius mode for final gather processing. Settings are On, Off, or View.

- **On**. Specifies that the setting means the Max Radius setting is used for final gather processing. The radius is specified in world units, and defaults to 10 percent of the maximum circumference of the model.
- **Off**. Specifies the maximum radius is the default value of 10 percent of the maximum model radius, in world units.
- **View**. Specifies the Max Radius setting in pixels instead of world units and is used for final gather processing.

**Max Radius** Sets the maximum radius within which final gathering is processed. Reducing this value can improve quality at a cost of increased rendering time.

**Use Min** Controls whether the Min Radius setting is used during final gather processing. When on, the minimum radius setting is used for final gather processing. When off, the minimum radius is not used.

**Min Radius** Sets the minimum radius within which final gathering is processed. Increasing this value can improve quality but increase rendering time.

### Light Properties

Affects how lights behave when calculating indirect illumination. By default, the energy and photon settings apply to all lights in a scene.

**Photons/Light** Sets the number of photons emitted by each light for use in global illumination. Increasing this value increases the accuracy of global illumination, but also increases the amount of memory used and the length of render time. Decreasing this value improves memory usage and render time, and can be useful for previewing global-illumination effects.

**Energy Multiplier** Multiplies the global illumination, indirect light, intensity of the rendered image.

### Visual

Helps you understand why the renderer is behaving in a certain way.

**Grid** Renders an image that shows the coordinate space of objects, the world, or camera.

- **Object**. Shows local coordinates (UVW). Each object has its own coordinate space.

- **World**. Shows world coordinates (XYZ). The same coordinate system applies to all objects.
- **Camera**. Shows camera coordinates, which appear as a rectangular grid superimposed on the view.

**Grid Size** Sets the size of the grid.

**Photon** Renders the effect of a photon map. This requires that a photon map be present. If no photon map is present, the Photon rendering looks just like the nondiagnostic rendering of the scene: the renderer first renders the shaded scene, then replaces it with the pseudocolor image.

- **Density**. Renders the photon map as it is projected into the scene. High density is displayed in red, and lower values render in increasingly cooler colors.
- **Irradiance**. Similar to the Density rendering, but shades the photons based on their irradiance. The maximum irradiance is rendered in red, and lower values render in increasingly cooler colors.

**BSP** Renders a visualization of the parameters used by the tree in the BSP ray-trace acceleration method. If a message from the renderer reports excessively large depth or size values, or if rendering seems unusually slow, this can help you locate the problem.

- **Depth**. Shows the depth of the tree, with top faces in bright red, and increasingly deep faces in increasingly cool colors.
- **Size**. Shows the size of leaves in the tree, with differently sized leaves indicated by different colors.

### Processing

**Tile Size** Determines the tile size for rendering. To render the scene, the image is subdivided into tiles. The smaller the tile size, the more image updates are generated during rendering. When the tile size is reduced, the number of image updates increases, meaning that a rendering take longer to complete. If the tile size is increased, fewer image updates occur and the rendering takes less time to complete.

**Tile Order** Specifies the method used (render order) for tiles as an image is rendered. You can choose a method based on how you prefer to see the image appear as it renders in the Render Window.

- **Hilbert**. Next tile to be rendered is based on the cost of switching to the next one.
- **Spiral**. Tiles are rendered beginning at the center of the image, and spiral outward.
- **Left to Right**. Tiles are rendered in columns, from bottom to top, left to right.
- **Right to Left**. Tiles are rendered in columns, from bottom to top, right to left.
- **Top to Bottom**. Tiles are rendered in rows, from right to left, top to bottom.
- **Bottom to Top**. Tiles are rendered in rows, from right to left, bottom to top.

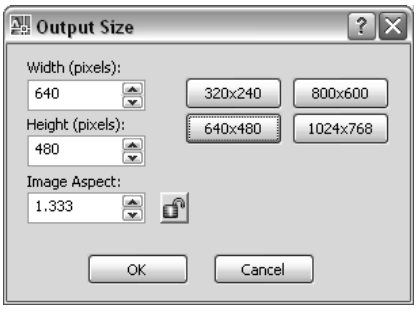
**Memory Limit** Determines the memory limit for rendering. The renderer keeps a count of the memory it uses at render time. If the memory limit is reached, the geometry for some objects is discarded in order to allocate memory for other objects.

# Output Size Dialog Box

## Quick Reference

**rpref**
Render panel (click icon to expand), Output Size, Specify Output Size

The Output Size dialog box is displayed when you choose *Specify Output Size* from the Output Size list. From the dialog box, you set the output resolution of the rendered image.




When you set a unique output size, it gets added to the Output Size list of the Render Settings palette. Four unique output sizes can populate the output size list, but they do not get saved with the current drawing and they are not maintained from one drawing session to the next.

### Width

Sets the width resolution of the output image, in pixels.

You can enter a new value in the width field either by entering the new value or by using the spinner to increase or decrease the value. You can set the width anywhere from 8 to 4096.

### Height

Sets the height resolution of the output image, in pixels.

You can enter a new value in the height field either by entering the new value or by using the spinner to increase or decrease the value. You can set the height anywhere from 8 to 4096.

### Image Aspect

Sets the aspect ratio, the ratio of width to height, of the rendered image.

Changing this value changes the Height value to maintain the correct dimensions for the active resolution. The image aspect value is always expressed as a multiplier value.

### Lock/Unlock Image Aspect

Locks the aspect ratio when using a custom output resolution.

When it is locked, the Width and Height spinners are locked to each other; adjusting one alters the other to maintain the aspect-ratio value. When

unlocked, changes to either Width or Height affects only the Image Aspect setting.

**NOTE** In viewports, the camera's frustum changes to reflect the image aspect ratio you set in the Output Resolution dialog box. This change takes place when you exit the dialog box.

#### Preset Resolution Buttons

Sets one of four most commonly used output resolutions.

# RPREFCLOSE

## Quick Reference

Closes the Render Settings palette if it is displayed

**rprefclose**

The RPREFCLOSE command closes the Render Settings palette (page 1226). If the Render Settings palette is currently displayed, either in an auto-hidden state or open state, it is closed.

# RSCRIPT

## Quick Reference

Repeats a script file

RSCRIPT is useful for demonstrations that repeat a script; for example, a script that must run over and over during a trade show or in a showroom.

If RSCRIPT is the last line in a script file, the file runs continuously until interrupted by ESC.

**NOTE** Consider turning off *UNDO* and any log files if you anticipate running the script over a long period; otherwise, these log files continue to grow and take up increasing amounts of disk space.

# RULESURF

## Quick Reference

Creates a ruled mesh between two curves

Draw ➤ Modeling ➤ Meshes ➤ Ruled MeshAt the Command prompt, enter rulesurf.

**rulesurf**
Current wire frame density: SURFTAB1=*current*
Select first defining curve:
Select second defining curve:

The objects you select define the edges of the ruled mesh. The objects can be points, lines, splines, circles, arcs, or polylines. If one of the boundaries is closed, then the other boundary must also be closed. You can use a point as the other boundary for either an open or a closed curve, but only one of the boundary curves can be a point. The 0,0 vertex is the endpoint of each curve nearest the point you used to select that curve.

For closed curves, the selection does not matter. If the curve is a circle, the ruled mesh begins at the 0-degree quadrant point, as determined by the current *X* axis plus the current value of the *SNAPANG* system variable. For closed polylines, the ruled mesh starts at the last vertex and proceeds backward along the segments of the polyline. Creating a ruled mesh between a circle and a closed polyline can be confusing. Substituting a closed semicircular polyline for the circle might be preferable.

examples of ruled surfaces

The ruled mesh is constructed as a 2 by *N* polygon mesh. RULESURF places half the mesh vertices at equal intervals along one defining curve, and the other half at equal intervals along the other curve. The number of intervals is specified by the *SURFTAB1* system variable. It is the same for each curve; therefore, the distance between the vertices along the two curves differs if the curves are of different lengths.

The *N* direction of the mesh is along the boundary curves. If both boundaries are closed, or if one is closed and the other is a point, the resulting polygon mesh is closed in the *N* direction and *N* equals SURFTAB1. If both boundaries

are open, *N* equals SURFTAB1 + 1, because division of a curve into *n* parts requires *n* + 1 tabulations.

Selecting objects at the same ends creates a polygon mesh.

Selecting objects at opposite ends creates a self-intersecting polygon mesh.

# S Commands

# 19

**In this chapter**

- SAVE
- SAVEAS
- SAVEIMG
- SCALE
- SCALELISTEDIT
- SCALETEXT
- SCRIPT
- SECTION
- SECTIONPLANE
- SECURITYOPTIONS
- SELECT
- SETBYLAYER
- SETIDROPHANDLER
- SETUV
- SETVAR
- SHADEMODE
- SHAPE
- SHEETSET
- SHEETSETHIDE
- SHELL
- SHOWMAT

- SIGVALIDATE
- SKETCH
- SLICE
- SNAP
- SOLDRAW
- SOLID
- SOLIDEDIT
- SOLPROF
- SOLVIEW
- SPACETRANS
- SPELL
- SPHERE
- SPLINE
- SPLINEDIT
- SPOTLIGHT
- STANDARDS
- STATUS
- STLOUT
- STRETCH
- STYLE
- STYLESMANAGER
- SUBTRACT
- SUNPROPERTIES
- SUNPROPERTIESCLOSE
- SWEEP
- SYSWINDOWS

# SAVE

## Quick Reference

Saves the drawing under the current file name or a specified name

**save**

The Save Drawing As dialog box (a standard file selection dialog box (page 931)) is displayed. Save the drawing under the current file name, or enter a different file name to save a copy of the drawing under that name.

See Standard File Selection Dialog Boxes (page 931) for a description of the options in this dialog box.

SAVE is available only from the command prompt. The Save option on the File menu or on the Standard toolbar is *QSAVE*. If the drawing is named, QSAVE saves the drawing without displaying the Save Drawing As dialog box. If the drawing is unnamed, the Save Drawing As dialog box is displayed. Enter a file name to both name and save the drawing.

# SAVEAS

## Quick Reference

Saves a copy of the current drawing under a new file name

File ➤ Save AsAt the Command prompt, enter saveas.
**saveas**

The Save Drawing As standard file selection dialog box (page 931) is displayed. Enter a file name and type. Saving a drawing to any DXF™ format affects performance.

---

**NOTE** AutoCAD 2004 is the drawing file format used by the AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006 releases.

---

See Save Drawings to Previous Drawing File Formats for a description of the limitations that result from saving to an earlier version.

In the Save Drawing As dialog box, Tools ➤ Options displays the Saveas Options dialog box (page 1243), which controls various DWG and DXF settings.

The file is saved with the specified file name. If the drawing is already named, the drawing is saved with the new file name. If you save the file as a drawing template, the Template Options dialog box (page 1242)is displayed, where you can provide a description for the template and set the units of measurement.

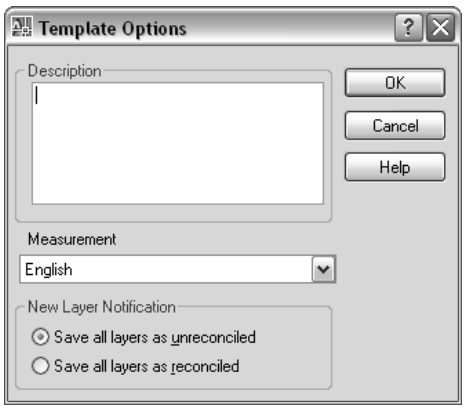
When *FILEDIA* is set to 0 (zero), SAVEAS displays command prompts (page 1245).

# Template Options Dialog Box

## Quick Reference

File ➤ Save AsAt the Command prompt, enter saveas.
**saveas**

Sets drawing template options.




**Description** Specifies a description for the drawing template. This description is displayed when you choose the template in the Startup dialog box or in the Create New Drawing dialog box (page 911).

**Measurement** Specifies whether the drawing template uses English or metric units.

### New Layer Notification

Controls whether the template file is saved with layers that are set as unreconciled or reconciled. Saving a template file with unreconciled layers

does not create a layer baseline so that when the drawing is first saved or plotted, a New Layer Notification bubble is not displayed.

**Save All Layers As Unreconciled** Saves the template file with its layers set as unreconciled, which means a layer baseline is not created. (See Reconcile New Layers for more information about unreconciled layers.) This option is checked by default.

**Save All Layers As Reconciled** Saves the template file with its layers set as reconciled, which results in creating a layer baseline. (See Reconcile New Layers for more information about reconciled layers).

# Saveas Options Dialog Box

## Quick Reference

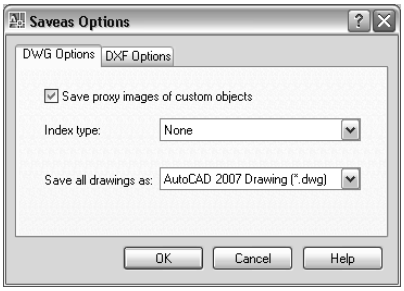
File ➤ Save As
**saveas**

Sets options for DWG files and for drawing interchange file output.

- DWG Options (page 1243)
- DXF Options (page 1244)

### DWG Options Tab (Saveas Options Dialog Box)

Controls the drawing index and display of custom objects and specifies the default file format for saving drawings.



### Save Proxy Images of Custom Objects

If you save to an earlier drawing file type, or the drawing contains custom objects from another application, you can select Save Proxy Images of Custom Objects to save images of the custom objects in the drawing file. If you do not choose this option, a frame is saved for each custom object in the drawing file. (*PROXYGRAPHICS* system variable)

**NOTE** If you are saving a drawing file that contains DWF references to an earlier drawing format, this switch has no effect on DWF underlays as they will not appear as proxy objects or as a frame.

### Index Type

Determines whether layer or spatial indexes are created when you save a drawing. The indexes are used to improve performance during demand loading. Using indexes may slightly increase the time required to save a drawing. If a partially open drawing does not already contain spatial and layer indexes, this option is not available.

**None** Creates neither layer nor spatial indexes when you save a drawing.

**Layer** Loads only layers that are on and thawed.

**Spatial** Loads only the portion of the drawing within a clipped boundary.

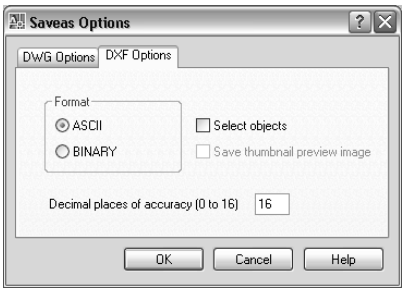
**Layer & Spatial** Optimizes performance by specifying that the program loads only layers that are on, thawed, and within a clipped boundary area.

### Save All Drawings As

Specifies the default file format that drawings are saved to. If you change the specified value, all subsequent uses of SAVE and QSAVE save the drawing to the new file format. You can also set this option on the Open and Save tab in the Options dialog box (see *OPTIONS*). Set the Save As option to AutoCAD 2007 Drawing to optimize performance while saving.

### DXF Options Tab (Saveas Options Dialog Box)

Sets drawing interchange file output options.




**Format** Specifies whether the program creates an ASCII or a binary DXF file. ASCII-format DXF files can be read with a text editor and are compatible with a wider range of applications. Binary-format DXF files contain all of the information of an ASCII DXF file but in a more compact form. You can read and write to binary-format files faster than to ASCII-format files.

For more information about DXF files, see the *DXF Reference* in the Help system.

**Select Objects** Controls whether the DXF file consists of selected objects or the entire drawing. When you select this option, the output file includes only selected objects and the block reference portions of any included blocks. The output file does not include the block definition tables.

**Save Thumbnail Preview Image** Specifies whether an image of the drawing is displayed in the Preview area of the Select File dialog box. Save Thumbnail Preview Image is also controlled by the *RASTERPREVIEW* system variable.

**Decimal Places of Accuracy** Saves the file using the specified number of bits of precision. The default precision is adequate in most cases; however, you might need to increase this value for certain drawings and for certain applications. If you encounter problems loading a DXF file you create, try increasing the precision. The only disadvantage of higher precision is increased file size. Lower precision is useful for specialized purposes but is not recommended in general.

# SAVEAS Command Line

## Quick Reference

When *FILEDIA* is set to 0 (zero), SAVEAS displays the following command prompts.

Current file format: *current*

Enter file format
[R14(LT98&LT97)/[2000(LT2000)/2004(LT2004)/2007(LT2007)/Standards/DXF/Template]
<2004>:
*Enter an option or press* ENTER
Save drawing as <*current*>: *Enter a name or press* ENTER

# SAVEIMG

## Quick Reference

Saves a rendered image to a file

Tools ➤ Display Image ➤ SaveAt the Command prompt, enter saveimg.
**saveimg**

The Save Rendered Image dialog box (page 1190) is displayed.

**NOTE** SAVEIMG is not available if the current rendering device does not support scan-line images.

# SCALE

## Quick Reference

Enlarges or reduces selected objects proportionally in the *X*, *Y*, and *Z* directions

Modify 
Modify ➤ ScaleDoes not exist in the menus.
Select the objects to scale, and right-click in the drawing area. Click Scale.
**scale**
Select objects: *Use an object selection method and press* ENTER *when you finish*
Specify base point: *Specify a point*

The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

**NOTE** When you use the SCALE command with objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.

Specify scale factor (page 1247) or [Copy (page 1247)/Reference (page 1247)]: *Specify a scale, enter* **c**, *or enter* **r**

**Scale Factor** Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects. You can also drag the cursor to make the object larger or smaller.

**Copy** Creates a copy of the selected objects for scaling.

**Reference** Scales the selected objects based on a reference length and a specified new length.

Specify reference length <1>: *Specify a beginning length from which to scale the selected objects*

Specify new length or [Points]: *Specify a final length to which to scale the selected objects, or enter* **p** *to define a length with two points.*

# SCALELISTEDIT

## Quick Reference

Controls the list of scales available for layout viewports, page layouts, and plotting

Format ➤ Scale ListAt the Command prompt, enter scalelistedit.
**scalelistedit** (or '**scalelistedit** for transparent use)

The Edit Scale List dialog box (page 1247) is displayed.

If you enter **-scalelistedit** at the command prompt, options are displayed at the command prompt (page 1251).

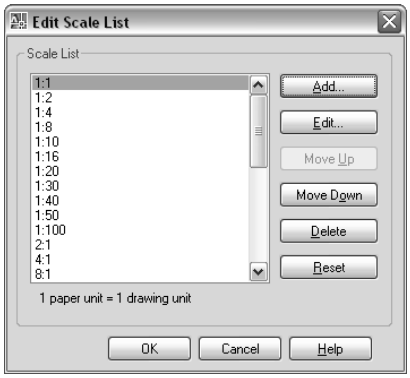
# Edit Scale List Dialog Box

## Quick Reference

Format ➤ Scale ListAt the Command prompt, enter scalelistedit.

**scalelistedit** (or '**scalelistedit** for transparent use)

Controls the list of scales available for layout viewports, page layouts, and plottting.




### Scale List

Displays the list of currently defined scales. Also displays temporary scales that are imported when xrefs are attached. If a scale name is duplicated, but has a different value, a number is appended to the name.

### Add

Displays the Add Scale dialog box (page 1249).

### Edit

Displays the Edit Scale dialog box (page 1250).

**NOTE** You cannot edit temporary scales.

### Move Up

Moves the currently selected scale in the scale list up one position.

### Move Down

Moves the currently selected scale in the scale list down one position.

### Delete

Removes the currently selected scale from the scale list.

**NOTE** You cannot delete a scale that is referenced by an object.

### Reset

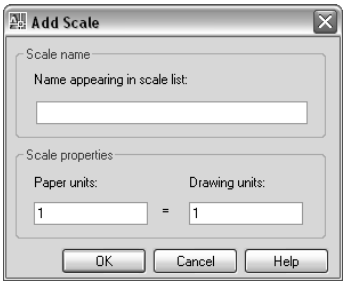
Deletes all custom scales and restores the default list of scales displayed in the scale list.

# Add Scale Dialog Box

## Quick Reference

Format ➤ Scale ListAt the Command prompt, enter scalelistedit.
**scalelistedit** (or '**scalelistedit** for transparent use)

Adds a new scale to the scale list.




### Scale Name

Specifies the name to appear in the scale list.

**Name Appearing in Scale List** Specifies the scale name to add to the scale list.

### Scale Properties

Sets the ratio of paper units to drawing units.

**Paper Units** When combined with the value of the drawing units, determines the ratio that defines the scale for viewing or plotting.

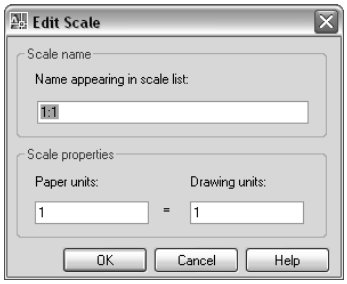
**Drawing Units** When combined with the value of the paper units, determines the ratio that defines the scale for viewing or plotting.

# Edit Scale Dialog Box

## Quick Reference

Format ➤ Scale ListAt the Command prompt, enter scalelistedit.
**scalelistedit** (or '**scalelistedit** for transparent use)

Changes the existing scales listed in the Scale List area.




### Scale Name

Lists the name of the currently selected scale in the Scale List area.

**NOTE** You cannot change the name of a scale that is referenced by an xref.

**Name Appearing in Scale List** Enter a descriptive or numeric name. For example, in an architectural drawing that uses a typical imperial scale, you can enter either 1"=4' or 1:48.

### Scale Properties

Modifies the ratio of paper units to drawing units.

**NOTE** You cannot change the scale properties of a scale that is referenced by an object in the drawing.

**Paper Units** When combined with the value of the drawing units, determines the ratio that defines the scale for viewing or plotting.

**Drawing Units** When combined with the value of the paper units, determines the ratio that defines the scale for viewing or plotting.

# -SCALELISTEDIT

## Quick Reference

If you enter **-scalelistedit** at the command prompt, the following SCALELISTEDIT command prompts are displayed.

Enter option [?/Add/Delete/Reset/Exit] <Add>: *Enter an option or press* ENTER

### ?

Displays a list of defined scales.

### Add

Adds a new scale to the scale list.

Enter name for new scale: *Enter a descriptive or numeric name such as 1"=4' or 1:48.*
Enter scale ratio: *Enter a ratio in the format n:m, where n is the number of paper units and m is the number of drawing units*

### Delete

Removes a specified scale. If a scale is current or supported by an object, it cannot be deleted.

Enter scale name to delete: *Enter the name of a scale that you want to delete*

### Reset

Deletes all custom scales as a result and restores the default list of scales.

Reset scale list to defaults? [Yes/No]: *Enter* **y** *or* **n**

### Exit

Exits the command.

# SCALETEXT

## Quick Reference

Enlarges or reduces selected text objects without changing their locations

Text 

Modify ➤ Object ➤ Text ➤ ScaleAt the Command prompt, enter scaletext.

**scaletext**

Select objects: *Use an object selection method, and press* ENTER *when you finish*

Enter a base point option for scaling

[Existing/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]<Existing>: *Specify a location to serve as a base point for scaling*

With the base point prompt, you choose one of several locations to serve as base points for scaling, which is used individually for each selected text object. The base point for scaling is established on one of several insertion point locations for text options, but even though the options are the same as when you choose an insertion point, the justification of the text objects is not affected.

The base point options shown above are described in the *TEXT* command. The base point options for single line text are similar to those for multiline text except that the Align, Fit, and Left text options are equivalent to the bottom left (BL) multiline text attachment point.

Specify new model height or [Paper height (page 1252)/Match object (page 1252)/Scale factor (page 1253)]<0.5000>: *Specify a text height or enter an option*

**NOTE** You can only specify a model height for non objects.

### Paper Height

Scales the text height depending on the annotative property.

**NOTE** You can only specify a paper height for annotative objects.

### Match Object

Scales the text objects that you originally selected to match the size of a selected text object.

**NOTE** This option only affects like objects (annotative or nonannotative).

Select a text object with the desired height: *Select a text object to match*

#### Scale Factor

Scales the selected text objects based on a reference length and a specified new length.

Specify scale factor or [Reference]: *Specify a scale factor or enter* **r**
**Scale Factor** Scales the selected text objects using the numeric scale factor that you enter.

**Reference** Scales the selected text objects relative to a reference length and a new length.

Specify reference length <1>: *Enter a length to serve as a reference distance*

Specify new length: *Enter another length in comparison to the reference length*

The selected text is scaled by a ratio of the values that you entered for the new length and the reference length. If the new length is less than the reference length, the selected text objects are reduced in size.

# SCRIPT

## Quick Reference

Executes a sequence of commands from a script file

Tools ➤ Run ScriptAt the Command prompt, enter script.
**script** (or '**script** for transparent use)

The Select Script File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the file name of a script to run that script.

When *FILEDIA* is set to 0 (zero), SCRIPT displays the following command prompt.

Enter script file name <*current*>:

# SECTION

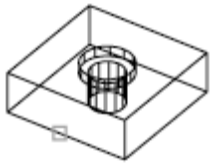
## Quick Reference

Uses the intersection of a plane and solids to create a region

**section**
Select objects: *Use an object selection method and press* ENTER *when you finish*

Selecting several solids creates separate regions for each solid.

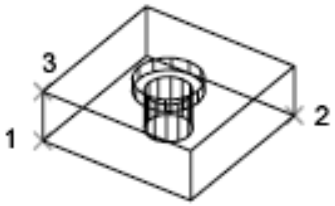
Specify first point (page 1254) on Section plane by [Object (page 1254)/Zaxis (page 1255)/View (page 1255)/XY (page 1255)/YZ (page 1256)/ZX (page 1256)] <3points>: *Specify a point or enter an option*



**First Point, 3 Points** Uses three points to define the sectioning plane. After you specify the first point, the following prompts are displayed:

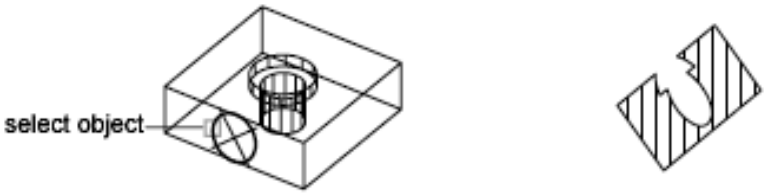
Specify second point on plane: *Specify a point (2)*

Specify third point on plane: *Specify a point (3)*




**Object** Aligns the sectioning plane with a circle, ellipse, circular or elliptical arc, 2D spline, or 2D polyline segment.

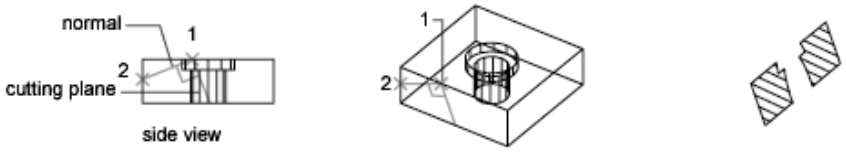
Select a circle, ellipse, arc, 2D-spline, or 2D-polyline:




**Z Axis** Defines the sectioning plane by specifying a point on the sectioning plane and another point on the plane's *Z* axis, or normal.

Specify a point on the section plane: *Specify a point (1)*

Specify a point on the Z-axis (normal) of the plane: *Specify a point (2)*

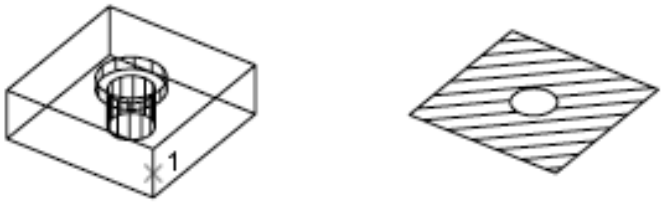



**View** Aligns the sectioning plane with the current viewport's viewing plane. Specifying a point defines the location of the sectioning plane.

Specify a point on the current view plane <0,0,0>: *Specify a point (1) or press* ENTER

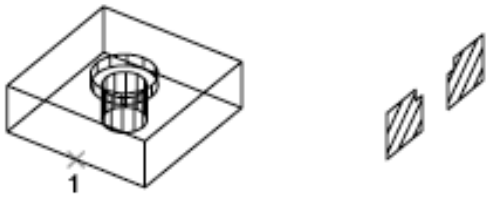

**XY** Aligns the sectioning plane with the *XY* plane of the current UCS. Specifying a point defines the location of the sectioning plane.

Specify a point on the XY-plane <0,0,0>: *Specify a point (1) or press* ENTER



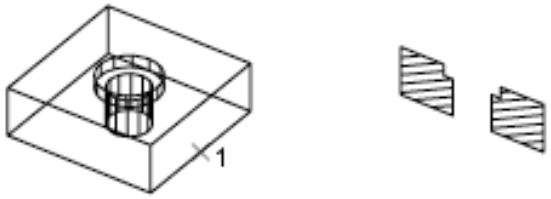
**YZ** Aligns the sectioning plane with the *YZ* plane of the current UCS. Specifying a point defines the location of the sectioning plane.

Specify a point on the YZ-plane <0,0,0>: *Specify a point (1) or press* ENTER




**ZX** Aligns the sectioning plane with the *ZX* plane of the current UCS. Specifying a point defines the location of the sectioning plane.

Specify a point on the ZX-plane <0,0,0>: *Specify a point (1) or press* ENTER




# SECTIONPLANE

## Quick Reference

Creates a section object that acts as a cutting plane through a 3D object

Draw ➤ Modeling ➤ Section PlaneAt the Command prompt, enter sectionplane.

**sectionplane**

3D Make panel, Section Plane

Selecting any point on the screen that is not on a face creates a section object independent of the solid. The first point establishes a point around which the section object rotates. The second point creates the section object.

Select a face or any point to locate the section line or [Draw section (page 1257)/Orthographic (page 1257)]: *Specify a point or enter an option*

Selecting a face on a solid or region aligns the section object parallel to that face.

Selecting any point on the screen that is not on a face creates a section object. The first point establishes a point around which the section object rotates. The second point creates the section object.

### Draw Section

Defines the section object with multiple points to create a section line with jogs.

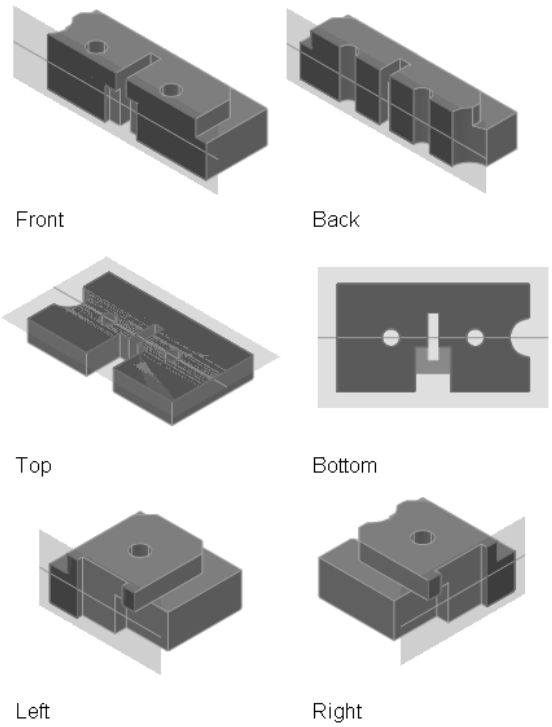
Specify start point: *Specify a point (1)*
Specify next point: *Specify a point (2)*
Specify next point or ENTER to complete: *Specify a point (3) or press ENTER.*
Specify next point in direction of section view: *Specify a point to indicate direction of the cutting plane.*

This option creates a section object in the Section Boundary state with live sectioning turned off.

### Orthographic

Aligns the section object to an orthographic orientation relative to the UCS.

Align section to: [Front/Back/Top/Bottom/Left/Right]: *Specify an option*



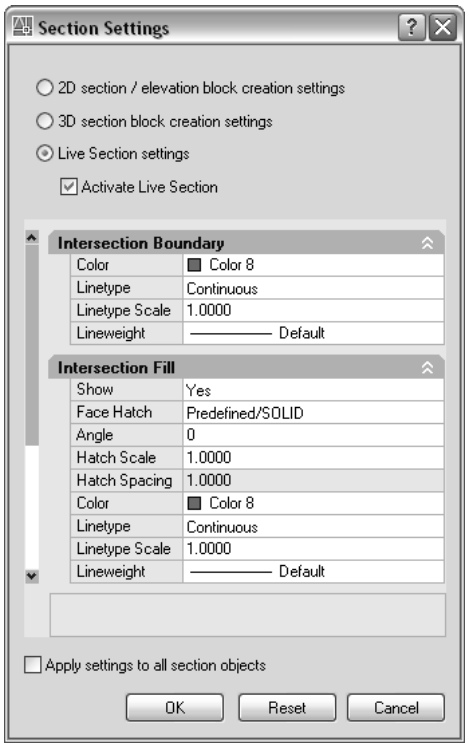

A section object is created with the specified orientation relative to the UCS (not the current view) and contains all 3D objects. This option creates a section object in the Section Plane state with live sectioning turned on.

## Section Settings Dialog Box

### Quick Reference

Right-click a section line. Click Live Section Settings.

The Section Settings dialog box contains display settings for creating 2D and 3D sections from the Generate Section / Elevation dialog box (page 1260) and for live sectioning. All settings are stored with the section object.




**2D Section / Elevation Block Creation Settings** Determines how a 2D section from a 3D object displays when generated.

**3D Section Block Creation Settings** Determines how a 3D object displays when generated.

**Live Section Settings** Determines how sectioned objects display in the drawing when live sectioning is turned on.

**Activate Live Section** Turns on live sectioning for the selected section object.

**Intersection Boundary** Sets the appearance of line segments that outline the intersection surface of the section object plane.

**Intersection Fill** Sets the optional fill that displays inside the boundary area of the cut surface where the section object intersects the 3D object.

**Background Lines** Controls the display of background lines for 2D and 3D sections. For 2D sections, also controls whether hidden lines are displayed.

**Foreground Lines** Controls the display of foreground lines.

**Curve Tangency Lines** Controls the inclusion of curved lines that are tangent to the section plane. Applies only to 2D sections.

**Static Image** Displays a description of the selected control.

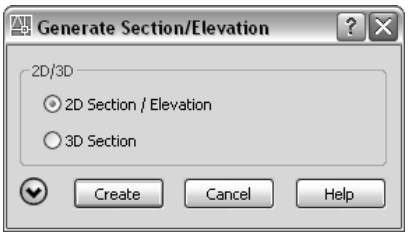
**Apply Settings to All Section Objects** When selected, applies all the settings to all section objects in the drawing. When clear, applies settings to the current section object only.

**Reset** Resets all settings in the dialog box to their default values.

# Generate Section / Elevation Dialog Box

## Quick Reference

Right-click a section line. Cick Generate 2D/3D Section.

**sectionplane** 


Creates 2D and 3D sections.

### 2D/3D

**2D Section/Elevation** Generates a 2D section.

**3D Section** Generates a 3D section.

### Source Geometry

**Include All Objects** Specifies to include all 3D objects (3D solids, surfaces, and regions) in the drawing, including those in xrefs and blocks.

**Select Objects to Include** Specifies that you will manually select the 3D objects (3D solids, surfaces, and regions) in the drawing from which to generate a section.

**Select Objects** Closes the dialog box temporarily while you select the objects for the section. When you finish selecting objects, press ENTER to redisplay the Generate Section/Elevation dialog box.

**Objects Selected** Indicates the number of objects selected.

#### Destination

**Insert as New Block** Inserts the generated section as a block in the current drawing.

**Replace Existting Block** Replaces an existing block in the drawing with the newly generated section.

Replaces an existing block in the drawing with the newly generated section.

**Select Block** Closes the dialog box temporarily while you select the section in the drawing. When you finish selecting the block, press ENTER to redisplay the Generate Section/Elevation dialog box.

**Export to a File** Saves the section to an external file.

**Filename and Path** Specifies a file name and path where the section will be saved.

#### Section Settings

Opens the Section Settings Dialog Box (page 1258).

#### Create

Creates the section.

# SECURITYOPTIONS

## Quick Reference

Controls security settings using the Security Options dialog box

#### securityoptions

The Security Options dialog box (page 1262) is displayed. You can add security settings that are applied when you save the drawing.

# Security Options Dialog Box

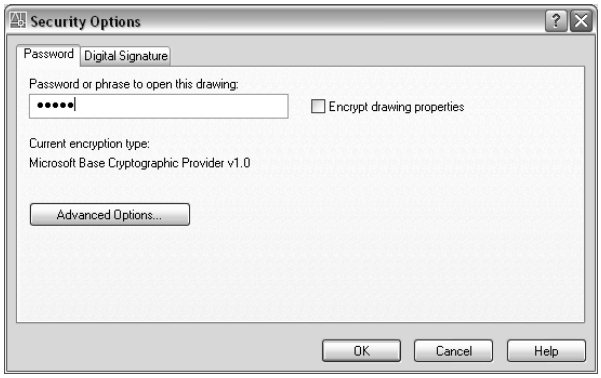
## Quick Reference

### securityoptions

Specifies security settings to be used when your drawing is saved.

### Password Tab (Security Options Dialog Box)

Adds a password to a drawing when the drawing is saved.




**Password or Phrase to Open This Drawing** Adds, changes, or removes a password the next time the drawing is saved. If you add or change the password, the Confirm Password dialog box (page 1263) is displayed. If you lose the password, it is not recoverable. Before you add a password, you should create a backup copy that is not protected with a password.

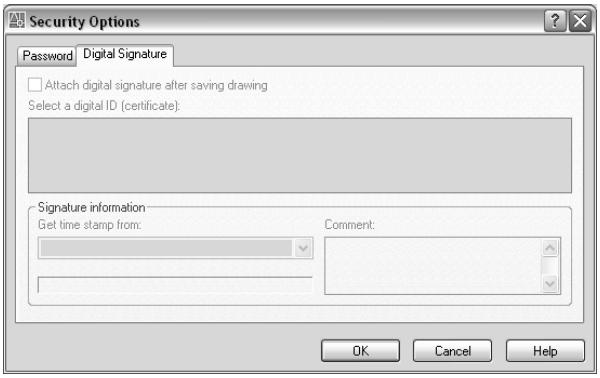
**Encrypt Drawing Properties** Encrypts drawing properties, so a password is required to view them. Drawing properties are details that help you identify the drawing, including title, author, subject, keywords that identify the model, or other important information.

**Current Encryption Type** Specifies the default encryption level supplied by your operating system, unless you choose an advanced level by clicking the Advanced Options button.

**Advanced Options** Opens the Advanced Options dialog box (page 1264), where you can choose an encryption provider and key length.

### Digital Signature Tab (Security Options Dialog Box)

Adds a digital signature to a drawing when the drawing is saved.




**Attach Digital Signatures After Saving Drawing** Attaches a digital signature to a drawing when the drawing is saved.

**Select a Digital ID (Certificate)** Displays a list of digital IDs that you can use to sign files. Includes information about the organization or individual to whom the digital ID was issued, the digital ID vendor who issued the digital ID, and when the digital ID expires.

**Signature Information** Provides a list of time services you can use to add a time stamp to your digital signature, the status of the time server connection, and a Comments area (to include information relevant to the digital signature or to the files you are signing).

**Get Time Stamp From** Provides a list of time servers you can use to time stamp your digital signature.

**Time Service Status** Displays the connection status (Successfully Contacted Time Server or Could Not Contact Time Server) of the time service.

**Comment** Provides a place for comments about the digital signature or the files you are signing.

# Confirm Password Dialog Box

## Quick Reference

**securityoptions**

Confirms the password entered in the Security Options dialog box (page 1262) that is added or changed the next time the drawing is saved.

**WARNING** If you lose the password, it is not recoverable. Before you add a password, you should create a backup that is not protected with a password.

## Advanced Options Dialog Box

### Quick Reference

**securityoptions**

Selects an encryption provider and key length for drawings that you protect with a password.

**Choose an Encryption Provider** Sets a level of encryption for a drawing. You can choose from encryption providers supplied by your operating system.

**Choose a Key Length** Sets a key length. The higher the key length, the higher the level of protection for your drawing.

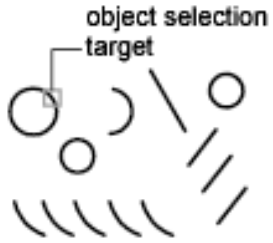
# SELECT

### Quick Reference

Places selected objects in the Previous selection set

**select**
Select objects: *Use an object selection method*

A small box, called the object selection target, replaces the crosshairs on the graphics cursor.



Objects must be selected in order to be processed. The Select Objects prompt occurs after many commands, including the SELECT command itself.

You can select objects individually with the pointing device, by drawing a selection window around them, by entering coordinates, or by using one of the selection methods listed below. These methods can be used to select objects regardless of the command that initiated the Select Objects prompt.

You can also press and hold the CTRL key to select original individual forms that are part of composite solids or vertices, edges, and faces on 3D solids. You can select one of these *subobjects*, or create a selection set of more than one subobject. Your selection set can include more than one type of subobject.

To view all options, enter **?** at the command prompt.

Expects a point or
Window (page 1265)/Last (page 1265)/Crossing (page 1265)/BOX (page 1266)/ALL (page 1266)/Fence (page 1266)/WPolygon (page 1267)/CPolygon (page 1267)/Group (page 1267)/Add (page 1267)/Remove (page 1268)/Multiple (page 1268)/Previous (page 1268)/Undo (page 1268)/AUto (page 1268)/SIngle (page 1268)/SUbobject (page 1268)/Object (page 1269)
Select objects: *Specify a point or enter an option*
**Window** Selects all objects completely inside a rectangle defined by two points. Specifying the corners from left to right creates a window selection. (Specifying the corners from right to left creates a crossing selection.)

Specify first corner: *Specify a point (1)*

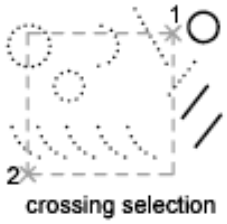
Specify opposite corner: *Specify a point (2)*


**Last** Selects the most recently created visible object. The object must be in the current space, that is, model space or paper space, and its layer must not be set to frozen or off.

**Crossing** Selects objects within and crossing an area defined by two points. A crossing selection is displayed as dashed or otherwise highlighted to differentiate it from window selection. Specifying the corners from right to left creates a crossing selection. (Specifying the corners from left to right creates a window selection.)

First corner: *Specify a point (1)*

Other corner: *Specify a point (2)*



crossing selection

**Box** Selects all objects inside or crossing a rectangle specified by two points. If the rectangle's points are specified from right to left, Box is equivalent to Crossing. Otherwise, Box is equivalent to Window.

Specify first corner: *Specify a point*

Specify opposite corner: *Specify a point*

**All** Selects all objects on thawed layers.


select all

**Fence** Selects all objects crossing a selection fence. The Fence method is similar to CPolygon except that that the fence is not closed, and a fence can cross itself. Fence is not affected by the *PICKADD* system variable.

First fence point: *Specify a point*

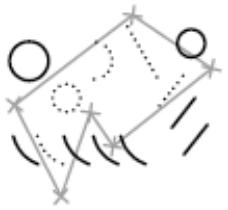
Specify endpoint of line or [Undo]: *Specify a point or enter* **u** *to undo the last point*


fence selection

**WPolygon** Selects objects completely inside a polygon defined by points. The polygon can be any shape but cannot cross or touch itself. The last segment of the polygon is drawn so that it is closed at all times. WPolygon is not affected by the *PICKADD* system variable.

First polygon point: *Specify a point*

Specify endpoint of line or [Undo]: *Specify a point or enter* **u** *to undo the last point*



WPolygon selection

**CPolygon** Selects objects within and crossing a polygon defined by specifying points. The polygon can be any shape but cannot cross or touch itself. The last segment of the polygon is drawn so that it is closed at all times. CPolygon is not affected by the *PICKADD* system variable.

First polygon point: *Specify a point*

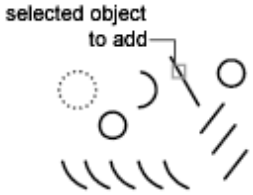
Specify endpoint of line or [Undo]: *Specify a point or enter* **u** *to undo the last point*

CPolygon selection

**Group** Selects all objects within a specified group.

Enter group name: *Enter a name list*

**Add** Switches to the Add method: selected objects can be added to the selection set by using any of the object selection methods. Auto and Add are the default methods.




**Remove** Switches to the Remove method: objects can be removed from the current selection set using any object selection method. An alternative to Remove mode is to hold down SHIFT while selecting single objects or use the Automatic option.

**Multiple** Specifies multiple points without highlighting the objects, thus speeding up the selection process for complex objects. The Multiple method also selects two intersecting objects if the intersection point is specified twice.

**Previous** Selects the most recent selection set. The Previous selection set is cleared by operations that delete objects from the drawing.

The program keeps track of whether each selection set was specified in model space or paper space. The Previous selection set is ignored if you switch spaces.

**Undo** Cancels the selection of the object most recently added to the selection set.

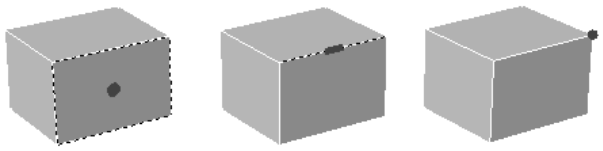
**Auto** Switches to automatic selection: pointing to an object selects the object. Pointing to a blank area inside or outside an object forms the first corner of a box defined by the Box method. Auto and Add are the default methods.

**Single** Switches to the Single method: selects the first object or set of objects designated rather than continuing to prompt for further selections.

**Subobject** Allows you to select original individual forms that are part of composite solids or vertices, edges, and faces on 3D solids. You can select one of these *subobjects*, or create a selection set of more than one subobject. Your selection set can include more than one type of subobject.

Select objects: *Select original individual forms that are part of composite solids or vertices, edges, and faces*

Pressing and holding the CTRL key is the same as selecting the SELECT command's Subobject option.



**Object** Ends the ability to select subobjects. Allows you to use object selection methods.

Select objects: *Use an object selection method*

# SETBYLAYER

## Quick Reference

Changes property and ByBlock settings for selected objects to ByLayer

Modify ➤ Change to ByLayer
**setbylayer**
Current active settings: Color Linetype Lineweight Material Plot Style
Select objects or [Settings]: *Use an object selection method and press* ENTER *when you finish*

If Settings is selected, the SetByLayer Settings dialog box (page 1269)is displayed, in which you can specify which object properties are set to ByLayer.

If objects are selected, the Command prompt displays:

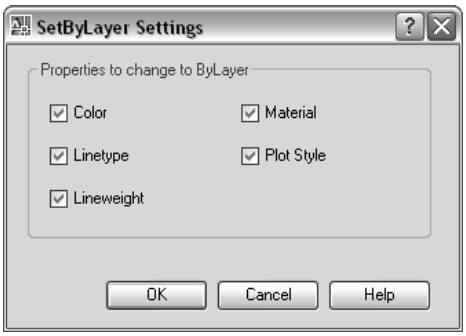
Change ByBlock to ByLayer? [Yes/No] <Yes>:
Include blocks? [Yes/No] <Yes>:

# SetByLayer Settings Dialog Box

## Quick Reference

Modify ➤ Change to ByLayer
**setbylayer**

Changes property overrides for color, linetype, lineweight, material, and plot style to ByLayer for selected objects and inserted blocks on unlocked layers.




**Color** Changes the color of selected objects to ByLayer. (SETBYLAYERMODE=1)

**Linetype** Changes the linetype of selected objects to ByLayer. (SETBYLAYERMODE=2)

**Lineweight** Changes the lineweight of selected objects to ByLayer. (SETBYLAYERMODE=4)

**Material** Changes the material of selected objects to ByLayer. (SETBYLAYERMODE=8)

**Plot Style** Changes the plot style of selected objects to ByLayer. This option is available in named plot style drawings. (SETBYLAYERMODE=16)

# SETIDROPHANDLER

## Quick Reference

Specifies the default type of i-drop content for the current Autodesk application

**setidrophandler**

The SETIDROPHANDLER command displays the Set Default i-drop Content Type dialog box (page 1271), where you set the default type of i-drop content for the Autodesk application you are currently working in. This setting determines the type of content that is inserted into your drawing when you drag a representative i-drop content image from the Web into your drawing.

## Set Default i-drop Content Type Dialog Box

### Quick Reference

**setidrophandler**

Specifies the default type of i-drop content accepted by the current Autodesk application. The list contains the available content types for installed Autodesk products that provide this i-drop functionality.

The default content type determines the type of content that is inserted in your drawing. For example, if you select Block, when you drag a representative i-drop content image from a web page, the inserted content is an AutoCAD block.

## i-drop Options Dialog Box

### Quick Reference

Right-click a representative i-drop content image and drag it from a web page into your current drawing, and then click Block.

Displays the source URL, the log file name, and the file name for the i-drop content that is currently being inserted in the drawing, and allows you to specify the associated data files to be transferred and the location for the files.

# SETUV

### Quick Reference

Obsolete

**setuv**

Starts *MATERIALMAP*.

# SETVAR

## Quick Reference

Lists or changes the values of system variables

Tools ➤ Inquiry ➤ Set VariableAt the Command prompt, enter setvar.
**setvar** (or '**setvar** for transparent use)
Enter variable name (page 1272) or [? (page 1272)] <*current*>: *Enter a variable name, enter* **?**, *or press* ENTER
**Variable Name** Specifies the name of the system variable you want to set.

Enter new value for *variable_name* <*current*>: *Enter a new value or press* ENTER

You can also change the value of system variables at the Command prompt by entering the name of the variable and its new value.

**?—List Variables** Lists all system variables in the drawing and their current settings.

Enter variable(s) to list <*>: *Enter a wild-card pattern or press* ENTER

# SHADEMODE

## Quick Reference

Starts the VSCURRENT command

**shademode**

Prompts for the *VSCURRENT* command are displayed.

If you enter **-shademode** at the Command prompt or use SHADEMODE in a script, SHADEMODE displays command prompts.

# -SHADEMODE

## Quick Reference

If you enter **-shademode** at the command prompt or use SHADEMODE in a script, the following SHADEMODE commands prompts are displayed.

Enter option [2D wireframe (page 1273)/3D wireframe (page 1273)/Hidden (page 1273)/Flat (page 1273)/Gouraud (page 1273)/fLat+edges (page 1273)/gOuraud+edges (page 1273)] <*current*>:
**2D Wireframe** Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible.

**3D Wireframe** Displays the objects using lines and curves to represent the boundaries. Material colors that you have applied to the objects are shown.

**Hidden** Displays the objects using 3D wireframe representation and hides lines representing back faces.

**Flat Shaded** Shades the objects between the polygon faces. The objects appear flatter and less smooth than Gouraud-shaded objects. Materials that you have applied to the objects show when the objects are flat shaded.

**Gouraud Shaded** Shades the objects and smooths the edges between polygon faces. This gives the objects a smooth, realistic appearance. Materials that you have applied to the objects show when the objects are Gouraud shaded.

**Flat Shaded, Edges On** Combines the Flat Shaded and Wireframe options. The objects are flat shaded with the wireframe showing through.

**Gouraud Shaded, Edges On** Combines the Gouraud Shaded and Wireframe options. The objects are Gouraud shaded with the wireframe showing through.

# SHAPE

## Quick Reference

Inserts a shape from a shape file that has been loaded using LOAD

**shape**
Enter shape name (page 1273) or [? (page 1274)]: *Enter a name, or enter* **?**
**Shape Name** Loads the shape.

Specify insertion point:

Specify height <*current*>: *Specify a height or press* ENTER

Specify rotation angle <0>: *Specify an angle or press* ENTER

If a shape belongs to an external reference (xref) attached to the current drawing, the shape file is identified as externally dependent. Externally dependent shapes cannot be used in the current drawing unless they are reloaded.

**?—List Shapes** Lists shapes and the files in which the shapes are defined. If you enter a question mark (**?**), the following prompt is displayed:

Enter shape name(s) to list <*>: *Enter a name list or press* ENTER

If you enter a name, the program lists the name of the file in which the shape definition exists and ends SHAPE. If you enter an asterisk (*****), the program lists shape names and ends SHAPE.

# SHEETSET

## Quick Reference

Opens the Sheet Set Manager

Standard 
Tools ➤ Palettes ➤ Sheet Set ManagerDoes not exist in the menus.
**sheetset**

Displays the Sheet Set Manager (page 1274).

# Sheet Set Manager

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set ManagerDoes not exist in the menus.
**sheetset**

The Sheet Set Manager organizes, displays, and manages *sheet sets*, a named collection of drawing sheets. Each *sheet* in a sheet set is a layout in a drawing (DWG) file.

The top of the Sheet Set Manager window contains a list box, called the *Sheet List control*, and several buttons. The buttons vary, depending on the selected tab.

- Sheet List (page 1275)
- Sheet Views (page 1278)
- Model Views (page 1279)

#### Sheet List Control

The Sheet List control displays the the name of the current sheet set, or, if no sheet sets are open, the Open option. The Sheet List control provides the following options for all tabs:

**Names of Open Sheet Sets** Lists all open sheet sets, if any. A check is displayed next to the current sheet set. The current sheet set is the open sheet set that is displayed in the Sheet Set Manager window.

**Recent** Displays a list of recently opened sheet sets.

**New Sheet Set** Starts the Create Sheet Set wizard.

**Open** Displays the Open Sheet Set standard file selection dialog box (page 931).

## Sheet List Tab

### Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

The Sheet List tab displays an ordered list of sheets. You can organize these sheets under headings, called *subsets*, that you create.

This tab has the following buttons:

#### Publish to DWF

Publishes selected sheets or a sheet set to a specified DWF file. Automatically uses settings specified in the PUBLISH (page 1123) command.

#### Publish

Displays a list of Publish options. The description for each option is listed in alphabetical order under Shortcut Menu Options below.

#### Sheet Selections

Displays a menu where you can save, manage, and restore sheet selections by name. This makes it easy to specify a group of sheets for a publish, transmit,

or archive operation. The description for each option is listed in alphabetical order under Shortcut Menu Options below.

### Details (at bottom of window)

Displays basic information about the currently selected sheet or subset.

### Preview (bottom of window)

Displays a thumbnail preview of the currently selected sheet.

### Shortcut Menu Options

The following options are displayed on shortcut menus for the Sheet List tab. For easier access, they are listed in alphabetical order. The options displayed in each shortcut menu depend on context: in the tree view, if you right-click the name of the sheet set, a different shortcut menu will display than if you right-click the name of a subset or the name of a sheet.

**Archive** Displays the Archive a Sheet Set dialog box (page 87).

**Close Sheet Set** Closes the current sheet set, removing the sheet set information displayed in the Sheet Set Manager window.

**New Sheet** Displays the New Sheet dialog box (page 1283).

**Rename & Renumber** Displays the Rename & Renumber Sheet dialog box (page 1284).

**eTransmit** Displays the Create Transmittal dialog box (page 541).

**Import Layout as Sheet** Displays the Import Layouts as Sheets dialog box (page 1285).

**Include Plot Stamp** Turns the plot stamp on or off for the selected sheet set, subset, or sheet.

**Plot Stamp Settings** Displays the Plot Stamp dialog box (page 1070), in which you can specify the information, such as drawing name and plot scale, that you want applied to the plot stamp.

**Insert Sheet List Table** Displays the Insert Sheet List Table dialog box (page 1286). Insert Sheet List Table is available only on the sheet set shortcut menu.

**Manage Page Setups** Displays the Page Setup Manager dialog box (page 1001).

**New Subset** Displays the Subset Properties dialog box (page 1281), where you can create a new sheet subset for organizing the sheets in a sheet set.

**Open** Opens the drawing file of the selected sheet and displays the layout.

**Open Read-Only** Opens the drawing file of the selected sheet in read-only mode and displays the layout. You cannot save changes to the file using the original file name.

**Properties (Sheet Set)** When the sheet set node is selected, displays the Sheet Set Properties dialog box (page 1289).

**Properties (Subset)** When a subset node is selected, displays the Subset Properties dialog box. See New Subset.

**Properties (Sheet)** When a sheet node is selected, displays the Sheet Properties dialog box (page 1293).

**Publish** Displays the Publish dialog box (page 1124).

**Publish in Reverse Order** When checked, sends sheets to the plotter in reverse of default order.

**Publish Using Page Setup Override** Automatically publishes the selected sheets using the selected page setup override rather than the page setup specified in each drawing. The page setup overrides are stored in a drawing template (DWT) file designated to be the source of the page setup override information.

If this item is not available, it means that there are no page setups specified, or that the DWT file is invalid or missing. Use the Manage Page Setups item on the same menu to determine the problem.

**Plot Stamp Settings** Displays the Plot Stamp dialog box (page 1070).

**Publish to DWF** Publishes the selected sheets or the sheet set to a specified DWF file. Automatically uses settings specified in the PUBLISH (page 1123) command.

**Publish to Plotter** Automatically publishes the selected sheets to the default plotter or printer.

**Resave All Sheets** Updates the sheet set information saved with each drawing in the current sheet set. Each drawing file in the current sheet set is opened and resaved. Any changes that were made are updated in the sheet set data (DST) file.

Drawing files saved in a previous DWG file format are resaved without changing format.

This operation is a background task and nothing is displayed in the program.

**NOTE** In a network environment, make sure that all drawing files in the current sheet set that are opened by other users are closed before performing this operation.

**Remove Subset** Removes the currently selected subset from the organization of the sheet set.

**Remove Sheet** Removes the currently selected sheet from the sheet set.

**Save Sheet Selection** Displays the New Sheet Selection dialog box (page 1294).

**Sheet Set Publish Options** Displays the Sheet Set Publish Options dialog box (page 1128). This is the same as the Publish Options dialog, but is specific to the current sheet set.

**Transmittal Setups** Displays the Transmittal Setups dialog box (page 544).

# Sheet Views Tab

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet Views tab)
**sheetset**

The Sheet Views tab displays an ordered list of views used in the current sheet set. You can organize these views under headings, called *categories*, that you create. You can display the list of views organized by categories or by the sheet on which they are located.

**NOTE** Only sheet views created in AutoCAD 2005 or later are listed on the Sheet Views tab.

This tab has the following buttons:

### New View Category Button

Displays the View Category dialog box (page 1295). This button is available only when the View by Category button is turned on and the views are displayed according to their categories.

### View by Category

Displays the views in the current sheet set organized by their categories.

### View by Sheet

Displays a list of views in the current sheet set organized by the sheet on which they are located.

### Shortcut Menu Options

The following options are displayed on shortcut menus for the Sheet Views tab. For easier access, they are listed in alphabetical order. The options displayed in each shortcut menu depend on context: if you right-click the sheet set, a different shortcut menu will display than if you right-click a view category, or a view.

**Display** Displays the selected view in the sheet in which it was created. Opens the drawing file containing the sheet, if the drawing file is not already open.

**New View Category** Displays the View Category dialog box (page 1295), in which you can create a new view category to organize the views in a sheet set.

**Place Callout Block** Specifies and places a callout block onto a sheet.

**Place View Label** Specifies and places a view label block onto a sheet.

**Properties (Sheet Set)** Displays the Sheet Set Properties dialog box (page 1289).

**Properties (View Category)** Displays the View Category dialog box (page 1295).

**Rename** Renames the selected sheet view category.

**Rename & Renumber** Displays the Rename and Renumber View dialog box (page 1285) where you can renumber and retitle the selected sheet view.

**Set Category** Reassigns the selected sheet view to a category that you specify.

## Model Views Tab

### Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Model Views tab)
**sheetset**

The Model Views tab displays a list of folders, drawing files, and model space views available for the current sheet set. You can add and remove folder locations to control which drawing files are associated with the current sheet set.

**NOTE** After creating a named model space view, you must save the drawing to add the view to the Model Views tab.

This tab has the following buttons:

### Refresh Button

Updates the list of drawing files listed in the tree view. The Refresh button updates all information stored in the sheet set data (DST) file, checks all folders in every resource location for new or removed drawing files, and checks all drawing files that are expanded for new or removed model space views. Alternatively, when the focus is on the Model Views tab, you can press F5 to perform a refresh.

### Add New Location Button

Displays the Browse for Folder standard file selection dialog box (page 931), in which you can add a folder location to the sheet set.

### Shortcut Menu Options

The following options are displayed on shortcut menus for the Model Views tab. For easier access, they are listed in alphabetical order. The options displayed in each shortcut menu depend on context: if you right-click a folder, a different shortcut menu will display than if you right-click a drawing within a folder.

**Add New Location** Displays the Browse for Folder standard file selection dialog box (page 931) where you can add a folder location to the sheet set.

**eTransmit** Displays the Create Transmittal dialog box (page 541).

**Open File** Opens the selected drawing (DWG) file.

**Place on Sheet** Initiates a series of automated steps that creates and places a view of the selected model onto the current layout of the current drawing. These steps are

- You are prompted to place the view on the layout. Right-click to change the scale of the view before placement
- The selected model is attached as an xref in the current drawing
- A layout viewport is created on the current layout
- A sheet view is created that corresponds with the newly created layout viewport

**Remove Location** Removes the currently selected folder location from the sheet set.

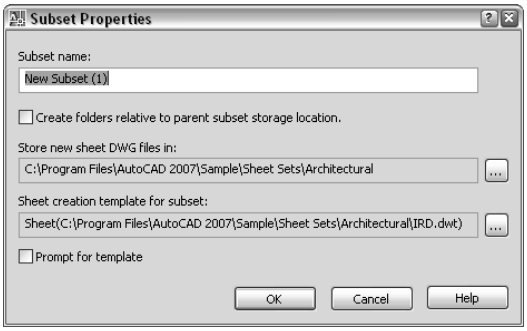
**See Model Space Views** Expands a list of named model space views.

# Subset Properties Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Creates a new sheet subset in a sheet set. You can organize sheets by dragging them to different subsets.




**Subset Name** Specifies the name of this subset and the name of the sheet storage folder for this subset if Create Folders Relative to Parent Subset Storage Location is selected.

**Create Folders Relative to Parent Subset Storage Location** When checked, creates a new folder under the parent folder when creating a new subset. This option provides a convenient method for creating a folder hierarchy in parallel with the subset hierarchy.

**Store New Sheet DWG Files In** Specifies the folder location for all new drawing (DWG) files containing sheets that are associated with this subset. The Browse button displays the Browse for Folder standard file selection dialog box (page 931).

**Sheet Creation Template for Subset** Specifies the drawing template (DWT) file and layout name that is used to create new drawing files for this subset.

The Browse button displays the Select Layout as Sheet Template dialog box (page 1282).

The syntax for this property is

*layoutname* [*folderpath*\]*filename.dwt*

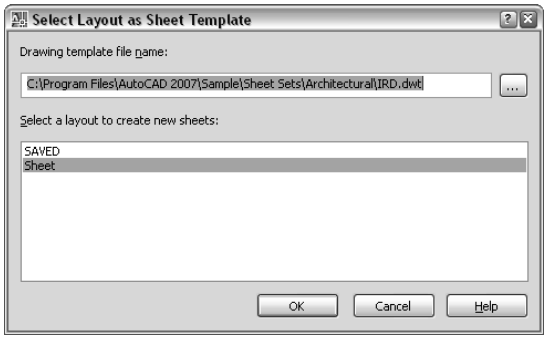
**Prompt for Template** When checked, prompts you to specify a drawing template file rather than use the default drawing template file for creating new sheets.

# Select Layout as Sheet Template Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Specifies the folder path, drawing template (DWT) file, and layout name to be used for creating new sheets.




**Drawing Template File Name** Specifies the drawing template file path and name to be used for creating new sheets. The Browse button displays the Select Drawing standard file selection dialog box (page 931).

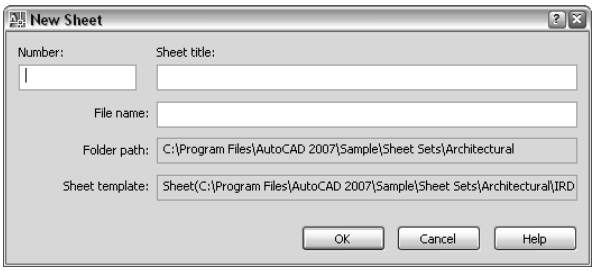
**Select a Layout to Create New Sheets** Lists and specifies the name of the layout in the drawing template file to be used for new sheets.

# New Sheet Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Creates a new sheet in the current sheet set by creating a new drawing (DWG) file that includes a layout tab with the same name as the drawing.




**Number** Specifies the sheet number. It is recommended that you do not fill in the sheet number when you first create a sheet. By default the drawing name and layout name of the new sheet will be the sheet number together with sheet title. You can remove the sheet number from the file name when you create the new sheet, but the sheet number will still be in the layout name.

**Sheet Title** Specifies the sheet title, which corresponds to an identically named layout tab in the new drawing.

**File Name** Specifies the name of the new drawing file containing the sheet. By default, the name of the drawing file is the sheet number combined with the sheet title. You can also change the drawing file name in this box.

**Folder Path** Displays the default folder for the new drawing file.

The default folder for a subset is specified in the Subset Properties dialog box (page 1281); and the default folder for a sheet set is specified in the Sheet Set Properties dialog box. (page 1289)

**Sheet Template** Displays the default sheet template file used for creating the new drawing file.

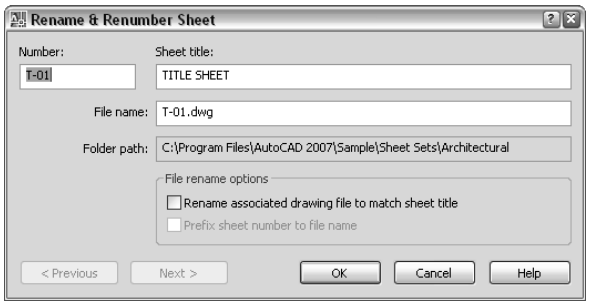
The default template file used for creating new drawings in a subset is specified in the Subset Properties dialog box (page 1281); the default template file for creating new drawings in a sheet set is specified in the Sheet Set Properties dialog box. (page 1289)

# Rename & Renumber Sheet Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Specifies a sheet number, sheet title, and other properties for a sheet in a sheet set.




**Number** Specifies the sheet number of the selected sheet.

**Sheet Title** Specifies the sheet title of the selected sheet.

**File Name** Specifies the name of the drawing file associated with the selected sheet.

**Folder Path** Displays the folder path for the drawing file.

**Rename Associated Drawing File to Match Sheet Title** When checked, changes the drawing file name to match the sheet title.

**Prefix Sheet Number to File Name** When checked, changes the drawing file name to a new name formed by adding the sheet number to the beginning of the sheet title.

**Next** Loads the next sheet into this dialog box. This provides a convenient method for renumbering or retitling a series of sheets.

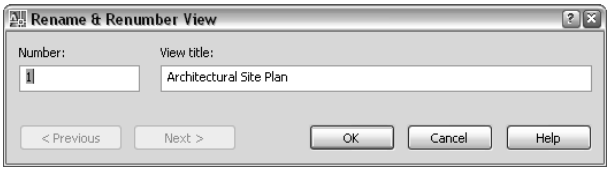
**Previous** Loads the previous sheet into this dialog box.

# Rename & Renumber View Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet Views tab)
**sheetset**

Specifies a sheet number and view title for a view in a sheet set.




**Number** Specifies the sheet number of the selected view.

**View Title** Specifies the view title of the selected view.

**Previous** Loads the previous view into this dialog box. This provides a convenient method for renumbering or retitling a series of views.

**Next** Loads the next view into this dialog box. This provides a convenient method for renumbering or retitling a series of views.

# Import Layouts as Sheets Dialog Box

## Quick Reference

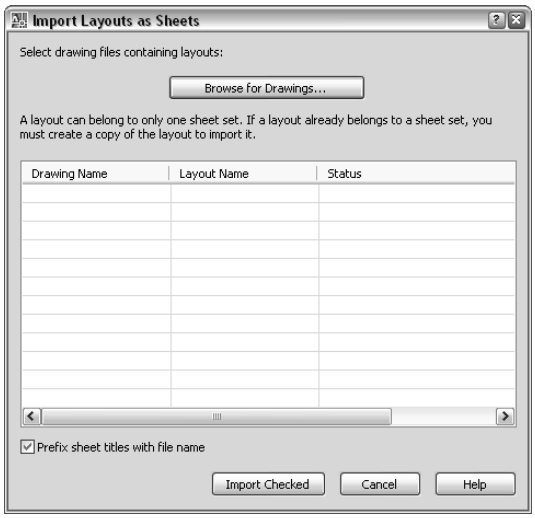
Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Provides a method for quickly importing layouts into a sheet set, and specifying which layout tab is to be used as the sheet.

---

**NOTE** A layout can belong to only one sheet set. If a layout already belongs to a sheet set, you must create a copy of the drawing containing the layout to import it.

---




**Browse for Drawings** Displays the Select Drawing standard file selection dialog box (page 931).

**List of Layouts in Selected Drawing** Lists all available layouts in the specified drawing file. Click a check box to select a layout.

**Prefix Sheet Titles with File Name** When checked, automatically adds the drawing file name to the beginning of the sheet title.

**Import Checked** Imports a layout from the list only if it displays a check mark.

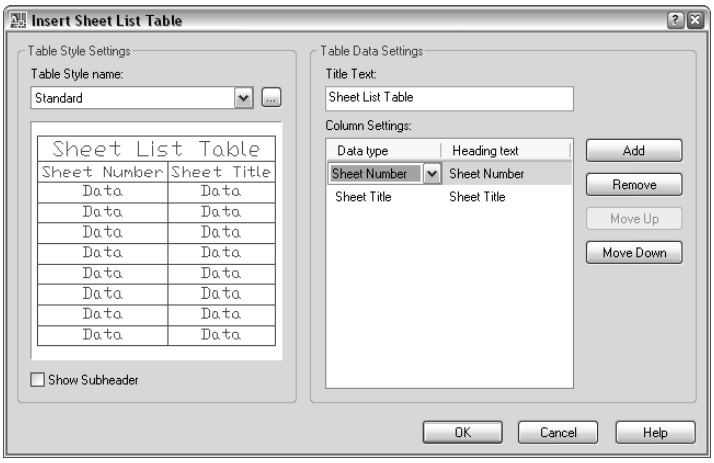
# Insert Sheet List Table Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Creates a table listing the sheets in the sheet set. This table can be added to any sheet in the current sheet set, but it is typically added to the title sheet.

**NOTE** Insert Sheet List Table on the sheet set shortcut menu is not available if the Model tab is active, or if the current layout is not a sheet in the current sheet set.




### Table Style Settings

Displays options for the table style.

**Table Style Name** Specifies the table style to be used for the table. The Browse button displays the Table Style dialog box (page 1402).

**Table Style Sample Area** Displays a sample of the currently selected table style.

**Show Subheader** When checked, displays the names of the subsets, dividing the sheet list table into sections.

### Table Data Settings

Displays options for the table data.

**Title Text** Specifies the name of the title of the sheet list table.

**Column Settings Area** Displays a list of the column definitions in the sheet list table. Each line in the list represents a column. The top-to-bottom order of the list represents how the columns will be displayed in the table (left to right).

**Data Type Column** Selecting a column definition and then clicking on an entry in the Data Type column displays a list. From this list, you can change the type of information that is going to be displayed in the columns of the sheet list table.

**Heading Text Column** Allows you to change the title text for each column in the sheet list table.

**Add** Adds a sheet number column to the sheet list table.

**Remove** Removes the selected column from the sheet list table. If you accidentally remove a column, you can add a new Sheet Number column and then change its data type from the data type list.

**Move Up** Moves the selected column up in the column list and to the left in the sheet list table.

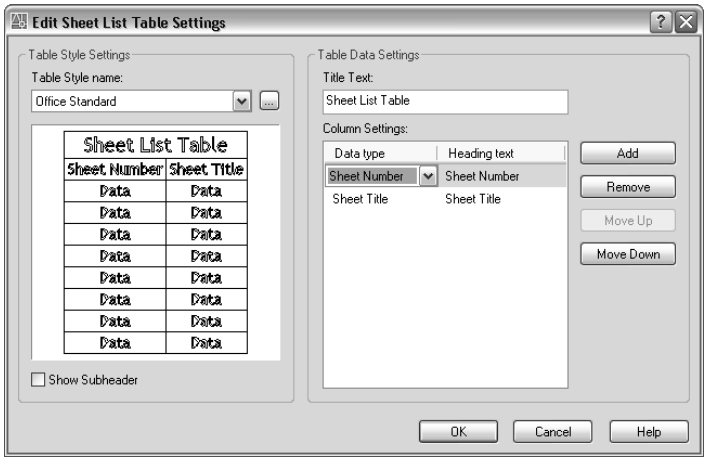
**Move Down** Moves the selected column down in the column list and to the right in the sheet list table.

# Edit Sheet List Table Settings Dialog Box

## Quick Reference

Select any cell in Sheet List Table ➤ Right-click to open the shortcut menu ➤ Edit Sheet List Table Settings

Allows you to edit a sheet list table that exists in the current drawing. The Edit Sheet List Table Settings dialog box gives you access to all the settings found on the Insert Sheet List Table dialog box (page 1286).




### Table Style Settings

Displays options for the table style.

**Table Style Name** Specifies the table style to be used for the table. The Browse button displays the Table Style dialog box (page 1402).

**Table Style Sample Area** Displays a sample of the currently selected table style.

**Show Subheader** When checked, displays the names of the subsets, dividing the sheet list table into sections.

#### Table Data Settings

Displays options for the table data.

**Title Text** Specifies the name of the title of the sheet list table.

**Column Settings Area** Displays a list of the column definitions in the sheet list table. Each line in the list represents a column. The top-to-bottom order of the list represents how the columns will be displayed in the table (left to right).

**Data Type Column** Selecting a column definition and then clicking on an entry in the Data Type column displays a list. From the list, you can change the type of information that is going to be displayed in the columns of the sheet list table.

**Heading Text Column** Allows you to change the title text for each column in the sheet list table.

**Add** Adds a sheet number column to the sheet list table.

**Remove** Removes the selected column from the sheet list table. If you accidentally remove a column, you can add a new Sheet Number column and then change its data type from the data type list.

**Move Up** Moves the selected column up in the column list and to the left in the sheet list table.

**Move Down** Moves the selected column down in the column list and to the right in the sheet list table.

## Sheet Set Properties Dialog Box

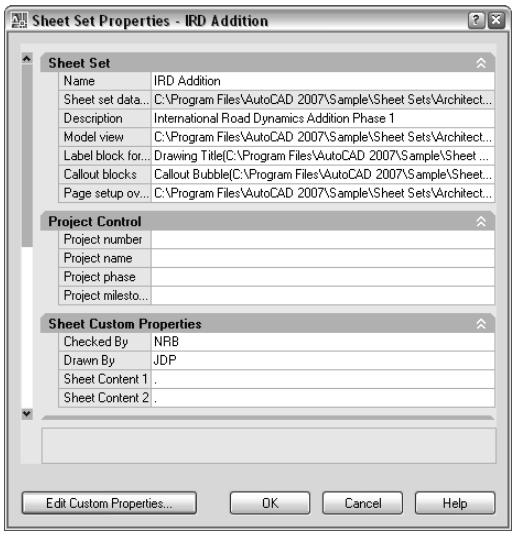
### Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)

**sheetset**

Displays information that is specific to the selected sheet set. You can click on each field to display a description at the bottom of the dialog box.

This includes information such as the path and file name of the sheet set data (DST) file, the paths of the folders that contain the drawing files associated with the sheet set, and any custom properties associated with the sheet set




**Name** Displays the name of the sheet set.

**Sheet Set Data File** Displays the path and file name of the sheet set data (DST) file.

**Description** Displays a description of the sheet set.

**Model View** Displays the paths and names of folders that contain drawings used by the sheet set.

**Label Block for Views** Displays the path and file name of the DWT or DWG file that contains the label blocks for the sheet set.

**Callout Blocks** Displays the path and file name of the DWT or DWG file that contains the callout blocks for the sheet set.

**Page Setup Overrides File** Displays the path and file name for the drawing template (DWT) file containing the page setup overrides for the sheet set.

**Project Control** Displays several fields commonly used in projects including Project Number, Project Name, Project Phase, and Project Milestone.

**Sheet Custom Properties** Displays the user-defined custom properties associated with each sheet in the sheet set.

**Sheet Storage Location** Displays the path and name of the folder where new sheets are created.

**Sheet Creation Template** Displays the path and name of the DWG or DWT file to be used when creating new sheets for the sheet set.

**Prompt for Template** Controls whether you will be prompted for a sheet creation template every time you create a new sheet in a sheet set.

**Sheet Set Custom Properties** Displays the user-defined custom properties associated with the sheet set.

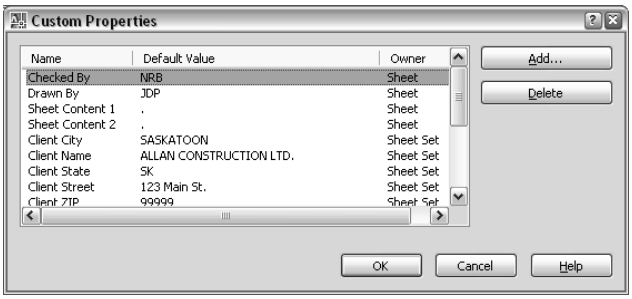
**Edit Custom Properties** Displays the Sheet Set Custom Properties dialog box (page 1291).

# Sheet Set Custom Properties Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Lists the custom properties associated with the current sheet set.




**Add** Displays the Add Custom Property dialog box (page 1292).

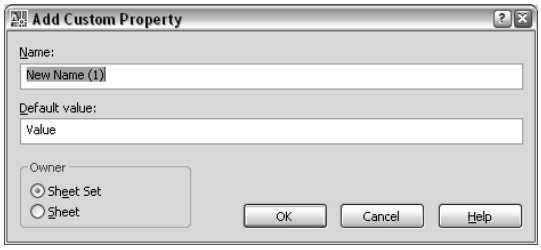
**Delete** Removes the selected custom property from the sheet set.

# Add Custom Property Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Creates a custom property to be associated with the current sheet set or with each sheet. Custom properties can be used to store information such as a contract number, the name of the designer, and the release date.




**Name** Specifies the name of a new custom property.

**Default Value** Specifies a value for the custom property.

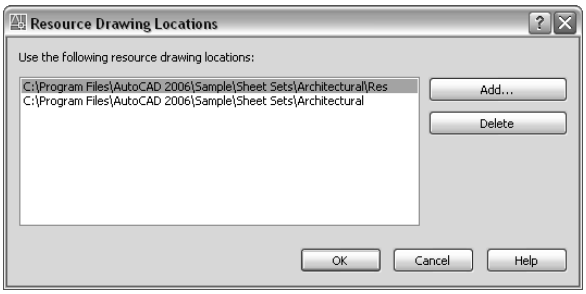
**Owner** Specifies whether the custom property belongs to the sheet set or to a sheet.

# Resource Drawing Locations Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager
**sheetset**

Displays a list of folders that are available for the current sheet set. You can add and remove folder locations to control which drawing files are available for the current sheet set.




**Add** Displays the Browse for Folder standard file selection dialog box (page 931), in which you can add a folder location to the list.

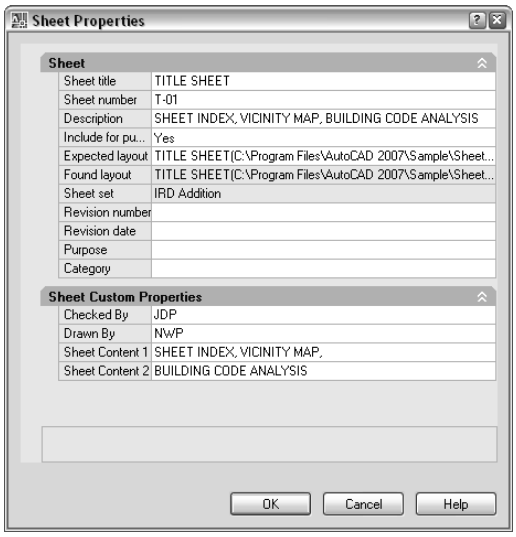
**Delete** Removes the selected folder from the list.

# Sheet Properties Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Displays information that is specific to the selected sheet, such as the sheet title, sheet number, and whether it will be included in publishing operations. Also displays custom properties, if any. You can enter a new value to modify any available sheet property.




**Sheet and Sheet Custom Properties** Displays the properties of the selected sheet. Also displays custom properties, if any. You can enter a new value to modify any sheet property. The Expected Layout is the path and file name of the drawing where the sheet was saved. The Found Layout is the path and file name of the drawing where the sheet was found. If the paths are different, you can modify the path and file name in Expected Layout. This operation reassociates the sheet with the sheet set.

# New Sheet Selection Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Saves a named selection of sheets for future operations. After you select several sheets and then save the selection, you can restore the sheet selection by name. This makes it easy to specify a set of sheets for a publish, transmit, or archive operation.

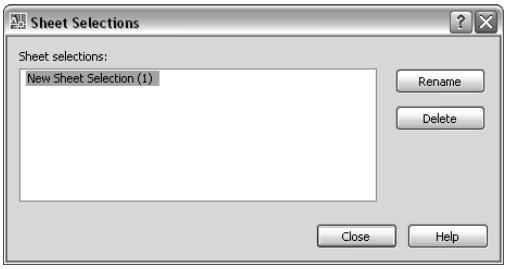

**Enter the Sheet Selection Name** Specifies a name for the sheet selection.

## Sheet Selections Dialog Box

### Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet List tab)
**sheetset**

Renames or deletes saved sheet selections.




**Sheet Selections** Specifies the sheet selection that you want to rename or delete.

**Rename** Renames the selected sheet selection.

**Delete** Deletes the selected sheet selection. Only the sheet selection is deleted. The sheets themselves are unaffected.

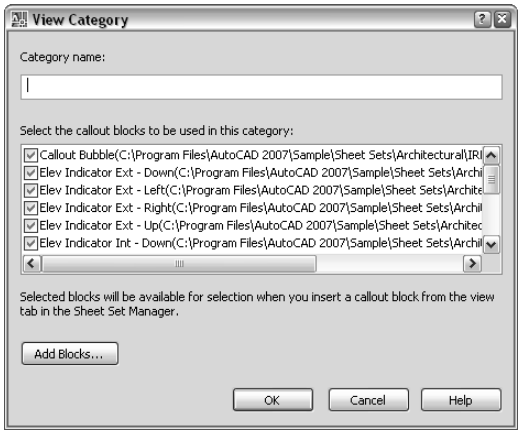
## View Category Dialog Box

### Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet Views tab)

**sheetset**

Creates a new view category in a sheet set. You can organize views by dragging them under different view categories.




**Category Name** Specifies the name of a new view category.

**Select the Callout Blocks to be Used in this Category** Provides a method for listing the callout blocks appropriate to the current view category.

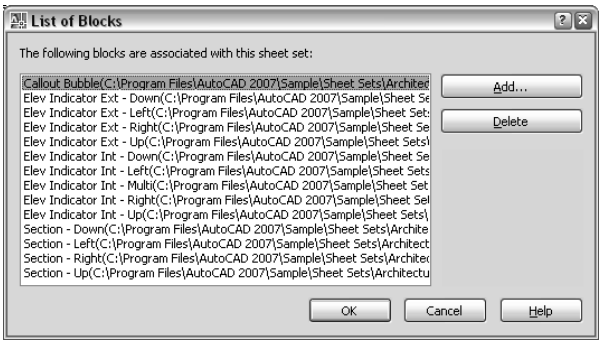
**Add Blocks** Displays the List of Blocks dialog box (page 1296).

# List of Blocks Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet Views tab)
**sheetset**

Lists, adds, or deletes callout blocks for use in the current sheet set. This dialog box is displayed by clicking the Add Blocks button in the View Category dialog box, or by clicking the [...] button next to Callout blocks in the Sheet Set Properties dialog box.




**List of Blocks** Displays the list of blocks available for use with the current sheet set.

**Add** Displays the Select Block dialog box (page 1297).

**Delete** Removes the selected block from the list of callout blocks.

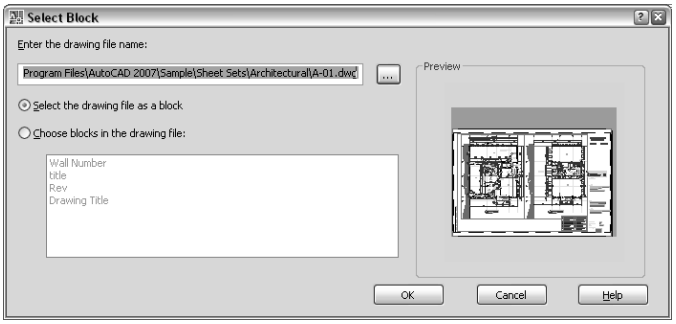
**Preview area** Displays a preview image when a single block is selected.

# Select Block Dialog Box

## Quick Reference

Tools ➤ Palettes ➤ Sheet Set Manager (Sheet Views tab)
**sheetset**

Adds a new label block or callout blocks for use in the current sheet set.




**Enter the Drawing File Name** Specifies the path and name of a drawing (DWG) file or the name and path of a drawing template (DWT) file to be used

as the source of a block definition. The Browse button displays Select Drawing, a standard file selection dialog box.

**Select the Drawing File as a Block** Uses the entire specified drawing file or drawing template file as the block definition.

**Choose Blocks in the Drawing File** Uses a selected block definition from the specified drawing file or drawing template file. When selecting callout blocks, you can use SHIFT or CTRL to select more than one block.

**Preview** Displays a preview image when a single block is selected.

# SHEETSETHIDE

## Quick Reference

Closes the Sheet Set Manager

Tools ➤ Palettes ➤ Sheet Set Manager
**sheetsethide**

Closes the Sheet Set Manager window.

# SHELL

## Quick Reference

Accesses operating system commands

**shell**
OS command: *Enter an operating system command or press* ENTER

With SHELL, you can execute operating system (OS) commands while remaining in this program. When SHELL prompts you for an OS command, you can enter most valid commands for your OS. When the command has been executed, SHELL returns you to the Command prompt.

Pressing ENTER at the OS Command prompt displays the system prompt with an extra close angle bracket (>). You can enter operating system commands as if you were at the normal system prompt. Enter **exit** to return to the Command prompt.

**WARNING** Do not use the SHELL command to delete lock files (file name extension *.??k*) or temporary files (file name extensions *.ac$* or *.$a*). Do not use SHELL to run chkdsk, reset the serial I/O ports, or run external programs that require disk swapping while editing a drawing stored on a floppy disk. Load Terminate-and-Stay-Resident programs into memory before starting this program.

# SHOWMAT

## Quick Reference

Obsolete

**showmat**

Starts *LIST*.

# SIGVALIDATE

## Quick Reference

Displays information about the digital signature attached to a file

**sigvalidate**

The Validate Digital Signatures dialog box (page 1299) is displayed. Review the information displayed in the dialog box, and click Close to view the signed file.

# Validate Digital Signatures Dialog Box

## Quick Reference

**sigvalidate**

Displays information about a digital signature. You can determine if a digital signature is valid and if the signed file has been modified since it was signed.


### Name

Displays the location and name of the file whose digital signature is being viewed.

### Digital Signature Status

Displays a Valid Signature icon if a digital signature is valid, and an Invalid Signature icon if the digital signature is not valid.

### File Status

Displays the status of the signed file.

### View Base Signature

Displays the Digital Signature Contents dialog box (page 1301). This option is available only if the digital signature is valid and the file has not been modified since it was signed.

### Xref Drawings

Contains detailed information about any xrefs in the signed file.

**View Xref Drawings** Displays the xref drawings contained in a file's base drawing.

**View Xref List** Contains a list of the xref drawings in a signed file. Displays the name, folder, and status of the xref. This list is available only if the current file contains xrefs and View Xref Drawings is selected.

**View Xref Signature** Displays the Digital Signatures Contents dialog box for the selected xref. This button is enabled only if the xref has a valid digital signature attached.

# Digital Signature Contents Dialog Box

## Quick Reference

**sigvalidate**

Displays information about a digital signature. You can determine if a digital signature is valid and if the signed file has been modified since it was signed.

### Name

Displays the location and name of the file whose digital signature is being viewed.

### Digital Signature Status

Displays the status of the digital signature.

### File Status

Displays the status of the signed file.

### Signed By

Displays the name of the organization or individual who attached a digital signature to the current file.

### Other Fields

Displays the subject, issuer, valid from date, valid to date, or serial number in the Values box.

**Subject** Displays information about the organization or individual who owns the digital ID and attached the digital signature.

**Issuer** Displays the name of the certificate authority that originally issued the digital ID.

**Valid From** Displays the day, month, year, and exact time from which the digital ID can be first used and is considered valid.

**Valid To** Displays the day, month, year, and exact time at which the digital ID ceases to be valid.

**Serial Number** Displays the serial number assigned to the digital ID.

#### Values

Displays information about a digital signature based on the item you select in the Other Fields list.

#### Comment

Displays any comments about the digital signature that is attached to the current file.

#### Signature Date and Time

Displays the date and time that the digital signature was attached to the current file. The date and time are based on the time service used when the signature was attached.

#### Time Service Used

Displays the time service used to add the time stamp to the current file.

#### Skip Xref Warnings

Determines if the Digital Signature Contents dialog box is displayed for signed xrefs. This check box is displayed only in the Digital Signature Contents dialog box when a signed drawing is opened.

# SKETCH

## Quick Reference

Creates a series of freehand line segments

**sketch**

The following information is based on the assumption that Tablet mode is on.

Record increment <*current*>: *Specify a distance or press* ENTER

The record increment value defines the length of the line segments. The pointing device must be moved a distance greater than the increment value to generate a line.


Sketching is captured as a series of independent lines. Setting the *SKPOLY* system variable to a nonzero value produces a polyline for each contiguous sequence of sketched lines rather than multiple line objects.

Sketch. Pen (page 1303)eXit (page 1303)Quit (page 1303)Record (page 1303)Erase (page 1303)Connect (page 1303). (page 1303) *Enter an option or press a pointer button*
**Pen (Pick Button)** Raises and lowers the sketching pen. The pen must be raised before you can select menu items with the pointing device.

**Exit—ENTER (button 3)** Records and reports the number of temporary lines sketched and ends the command.

**Quit (button 4)** Discards all temporary lines sketched since the start of SKETCH or the last use of the Record option, and ends the command.

**Record (button 2)** Records temporary lines as permanent and does not change the pen's position. Reports the number of lines using the following prompt:

nnn lines recorded.

**Erase (button 5)** Erases any portion of a temporary line and raises the pen if it is down.

Select end of delete.

**Connect (button 6)** Lowers the pen to continue a sketch sequence from the endpoint of the last sketched line or last Erase.

Connect: Move to endpoint of line.

**. (Period) (button 1)** Lowers the pen, draws a straight line from the endpoint of the last sketched line to the pen's current location, and returns the pen to the up position.

# SLICE

## Quick Reference

Slices a solid with a plane or surface

Modify ➤ 3D Operations ➤ SliceAt the Command prompt, enter slice.
**slice**
3D Make panel (click icon to expand), Slice
Select objects to slice: *Use an object selection method and press* ENTER *when you finish*

**NOTE** If you include regions in the selection set of objects to slice, they are ignored.

Specify start point of slicing plane or [planar Object (page 1304)/Surface (page 1305)/Zaxis (page 1305)/View (page 1306)/XY (page 1306)/YZ (page 1307)/ZX (page 1308)/3points (page 1308)] <3points>: *Specify a point, enter an option, or press* ENTER *to use the 3 Points option*
Specify second point on plane: *Specify a point*

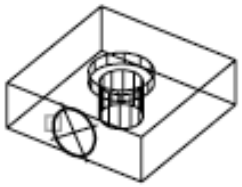
These two points define the angle of the slicing plane. The slicing plane is perpendicular to the current UCS.

Select solid to keep or [keep Both sides] <Both>: *Select one of the resulting solids or enter* **b**

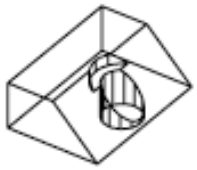
### Planar Object

Aligns the cutting plane with a circle, ellipse, circular or elliptical arc, 2D spline, or 2D polyline segment.

Select a circle, ellipse, arc, 2D-spline, or 2D-polyline:



object cutting plane



sliced object

You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids. However,

the resulting solids don't retain a history of the original forms that created them.

Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

### Surface

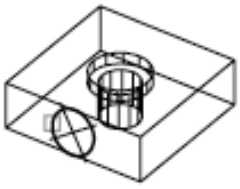
Aligns the cutting plane with a surface.

Select a surface:

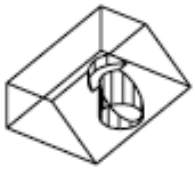
---

**NOTE** You cannot select meshes created with the *EDGESURF*, *REVSURF*, *RULESURF*, and *TABSURF* commands.

---



object cutting plane



sliced object

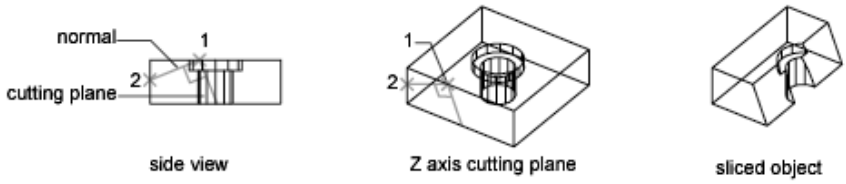
You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids. However, the resulting solids don't retain a history of the original forms that created them.

Select solid to keep or [keep Both sides] <Both>: *Select one of the resulting solids or enter* **b**

### Z Axis

Defines the cutting plane by specifying a point on the plane and another point on the *Z* axis (normal) of the plane.

Specify a point on the section plane: *Specify a point (1)*
Specify a point on the Z-axis (normal) of the plane: *Specify a point (2)*




You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids.

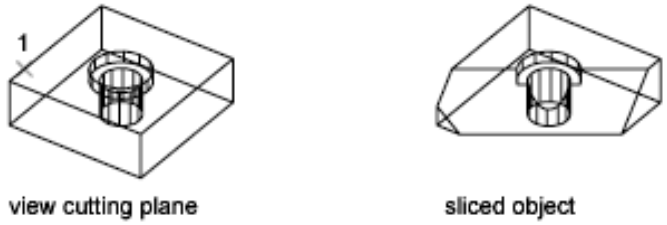
Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

### View

Aligns the cutting plane with the current viewport's viewing plane. Specifying a point defines the location of the cutting plane.

Specify a point on the current view plane <0,0,0>: *Specify a point (1) or press* ENTER




You can retain both halves of the sliced solid or just the half you specify. The sliced solids retain the layer and color properties of the original solid.

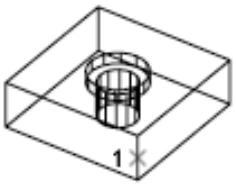
Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

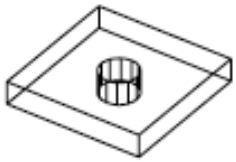
### XY

Aligns the cutting plane with the *XY* plane of the current user coordinate system (UCS). Specifying a point defines the location of the cutting plane.

Specify a point on the XY-plane <0,0,0>: *Specify a point (1) or press* ENTER




XY cutting plane



sliced object

You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids.

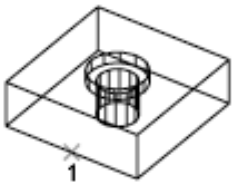
Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

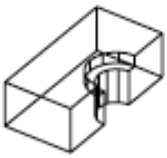
### YZ

Aligns the cutting plane with the *YZ* plane of the current UCS. Specifying a point defines the location of the cutting plane.

Specify a point on the YZ-plane <0,0,0>: *Specify a point (1) or press* ENTER




YZ cutting plane



sliced object

You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids.

Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

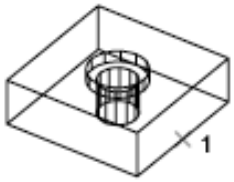
The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

### ZX

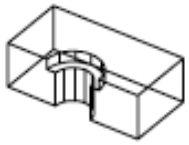
Aligns the cutting plane with the *ZX* plane of the current UCS. Specifying a point defines the location of the cutting plane.

Specify a point on the ZX-plane <0,0,0>: *Specify a point (1) or press* ENTER

If a single solid is sliced into more than two objects, one solid is created from the objects on one side of the plane and another solid is created from the objects on the other side.

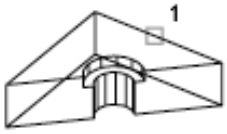



ZX cutting plane



sliced object

You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids.



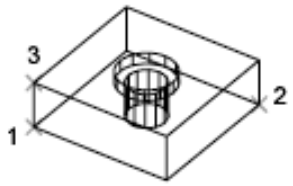

Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

The descriptions of the Point on the Desired Side and Keep Both Sides options match the descriptions of the corresponding options under 3 Points.

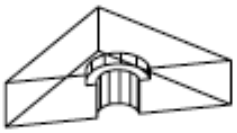
### 3 Points

Defines the cutting plane using three points.

Specify first point on plane: *Specify a point (1)*
Specify second point on plane: *Specify a point (2)*
Specify third point on plane: *Specify a point (3)*




3-point cutting plane

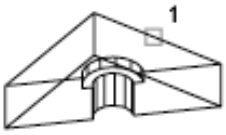


sliced object

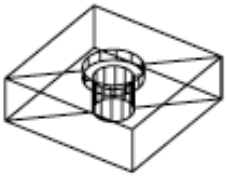
You can retain both halves of the sliced solids or just the half you specify. The sliced solids retain the layer and color properties of the original solids.

Specify a point on desired side or [keep Both sides] <Both>: *Specify a point or enter* **b**

**Point on the Desired Side** Uses a point to determine which side of the sliced solids your drawing retains. The point cannot lie on the cutting plane.




**Keep Both Sides** Retains both sides of the sliced solids. Slicing a single solid into two pieces creates two solids from the pieces on either side of the plane. SLICE never creates more than two new composite solids for each selected solid.



# SNAP

## Quick Reference

Restricts cursor movement to specified intervals

Status bar ➤ Snap
**snap** (or '**snap** for transparent use)

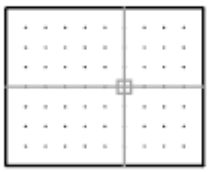
Specify snap spacing (page 1310) or [ON (page 1310)/OFF (page 1310)/Aspect (page 1310)/Style (page 1311)/Type (page 1312)] <*current*>: *Specify a distance, enter an option, or press* ENTER

### Snap Spacing

Activates Snap mode with the value you specify.

### On

Activates Snap mode using the current settings of the snap grid.



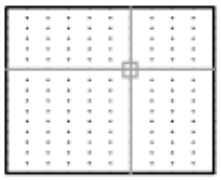
### Off

Turns off Snap mode but retains the current settings.

### Aspect

Specifies different spacing in the *X* and *Y* directions. This option is not available if the current snap style is Isometric.

Specify horizontal spacing <*current*>: *Specify a distance, or press* ENTER
Specify vertical spacing <*current*>: *Specify a distance, or press* ENTER



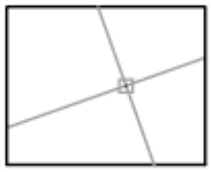
### Rotate (obsolete)

Sets the origin and rotation of the snap grid.

**NOTE** It is recommended that you use the UCS to control the grid origin and rotation instead.

The rotation angle is measured relative to the current UCS. You can specify a rotation angle between –90 and 90 degrees. A positive angle rotates the grid counterclockwise about its base point. A negative angle rotates the grid clockwise.

Specify base point <*current*>: *Specify a point, or press* ENTER
Specify rotation angle <*current*>: *Specify an angle distance, or press* ENTER



### Style

Specifies the format of the snap grid, which is Standard or Isometric.

Enter snap grid type [Standard/Isometric] <*current*>: *Enter* **s**, *enter* **i**, *or press* ENTER

### Standard

Sets a rectangular snap grid that is parallel to the *XY* plane of the current UCS. *X* and *Y* spacing may differ.

Specify snap spacing or [Aspect] <*current*>: *Specify a distance, enter* **a**, *or press* ENTER
**Spacing** Specifies the overall spacing of the snap grid.

**Aspect** Specifies the horizontal and vertical spacing of the snap grid separately.

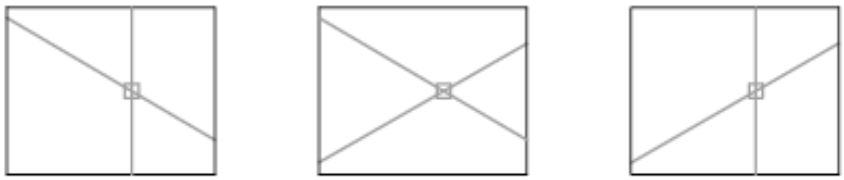
Specify horizontal spacing <*current*>: *Specify a distance or press* ENTER

Specify vertical spacing <*current*>: *Specify a distance or press* ENTER

### Isometric

Sets an isometric snap grid, in which the snap locations are initially at 30-degree and 150-degree angles. Isometric snap cannot have different Aspect values. The lined grid does not follow the isometric snap grid.

Specify vertical spacing <*current*>: *Specify a distance or press* ENTER




ISOPLANE (page 683) determines whether the crosshairs lie in the top isometric plane (30- and 150-degree angles), the left isoplane (90- and 150-degree angles), or the right isoplane (30- and 90-degree angles).

### Type

Specifies the snap type, polar or rectangular. This setting is also controlled by the SNAPTYPE (page 1815) system variable.

Enter snap type [Polar/Grid] <*current*>: *Enter* **p** *or* **g**
**Polar** Sets the snap to polar tracking angles that are set in the POLARANG (page 1785) system variable.

**Grid** Sets the snap to Grid. When you specify points, the cursor snaps along vertical or horizontal grid points.

# SOLDRAW

## Quick Reference

Generates profiles and sections in viewports created with SOLVIEW

Draw ➤ Modeling ➤ Setup ➤ DrawingAt the Command prompt, enter soldraw.
**soldraw**
Select viewports to draw ...
Select objects: *Select the viewports to be drawn*

SOLDRAW can only be used in viewports that have been created with *SOLVIEW*.

Visible and hidden lines representing the silhouette and edges of solids in the viewport are created and then projected to a plane perpendicular to the viewing direction. Silhouettes and edges are generated for all solids and portions of solids behind the cutting plane. For sectional views, cross-hatching is created

using the current values of the *HPNAME*, *HPSCALE*, and *HPANG* system variables.

Any existing profiles and sections in the selected viewport are deleted, and new ones are generated. All layers, except those required to display the profile or section, are frozen in each viewport.

**WARNING** Do not place permanent drawing information on the *view name*-VIS, *view name*-HID, and *view name*-HAT layers. The information stored on these layers is deleted and updated when SOLDRAW is run.

To undo a viewport drawn by SOLDRAW, you must use the Back option of *UNDO*.

**NOTE** SOLDRAW is defined by the *acsolids.arx* application and is intended to be used only interactively. For information about using this command from an application, see "Externally Defined Commands" in the *AutoLISP Reference*.

# SOLID

## Quick Reference

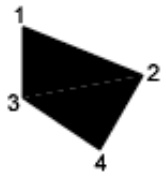
Creates solid-filled triangles and quadrilaterals

Draw ➤ Modeling ➤ Meshes ➤ 2D SolidAt the Command prompt, enter solid.

**solid**
Specify first point: *Specify a point (1)*
Specify second point: *Specify a point (2)*

The first two points define one edge of the polygon.

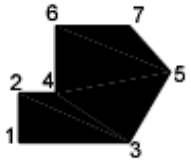
Specify third point: *Specify a point (3) diagonally opposite the second*
Specify fourth point or <exit>: *Specify a point (4) or press* ENTER



Pressing ENTER at the Fourth Point prompt creates a filled triangle. Specifying a point (5) creates a quadrilateral area.

The last two points form the first edge of the next filled area. The Third Point and Fourth Point prompts are repeated. Specifying successive third and fourth points creates further connected triangles and four-sided polygons in a single solid object. Pressing ENTER ends SOLID.

2D solids are filled only when the *FILLMODE* system variable is on and the viewing direction is orthogonal to the 2D solid.




# SOLIDEDIT

## Quick Reference

Edits faces and edges of 3D solid objects

Modify ➤ Solid Editing
**solidedit**
Solids editing automatic checking: SOLIDCHECK=*current*
Enter a solids editing option
[Face (page 1314)/Edge (page 1325)/Body (page 1327)/Undo (page 1329)/eXit (page 1329)]
<eXit>: *Enter an option or press* ENTER

### Face

Edits 3D solid faces by extruding, moving, rotating, offsetting, tapering, deleting, copying, or changing the color of the selected faces.

Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit]
<eXit>: *Enter an option or press* ENTER

### Extrude

Extrudes selected planar faces of a 3D solid object to a specified height or along a path. You can select multiple faces at one time.

Select faces or [Undo/Remove]: *Select one or more faces or enter an option*
Select faces or [Undo/Remove/ALL]: *Select one or more faces or enter an option*

### Undo

Cancels the selection of the faces you added most recently to the selection set and redisplays the prompt. If all faces have been removed, the following prompt is displayed.

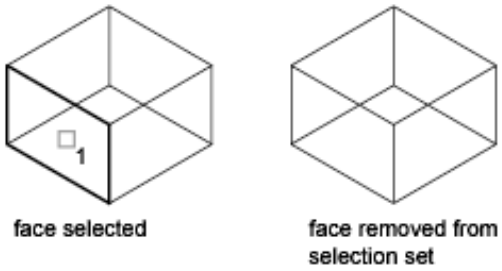
Face selection has been completely undone

### Remove

Removes previously selected faces from the selection set. The following prompt is displayed.

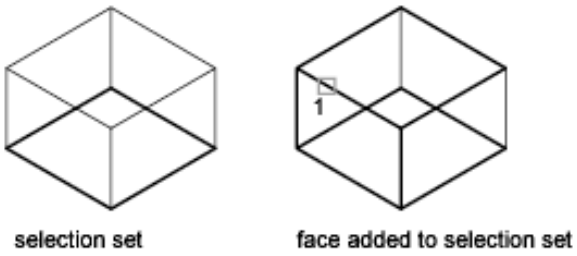
Remove faces or [Undo/Add/ALL]: *Select one or more faces (1), enter an option, or press* ENTER
**Undo** Cancels the selection of the faces you removed most recently from the selection set and redisplays the prompt. If no faces are currently selected, the following prompt is displayed.

Face selection has been completely undone




**Add** Adds faces to the selection set.

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1) or select an option*




- *Undo:* Cancels selection of the faces you added most recently to the selection set and redisplays the prompt.
- *Remove:* Removes previously selected faces and redisplays the prompt.
- *All:* Selects all faces and adds them to the selection set and redisplays the prompt.

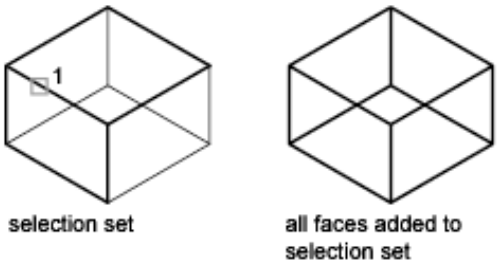
**All** Selects all faces and adds them to the selection set.

### All

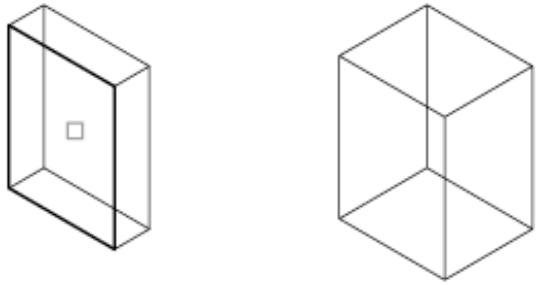
Selects all faces and adds them to the selection set.

After you select faces or choose an option, the following prompt is displayed.

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1), enter an option, or press* ENTER



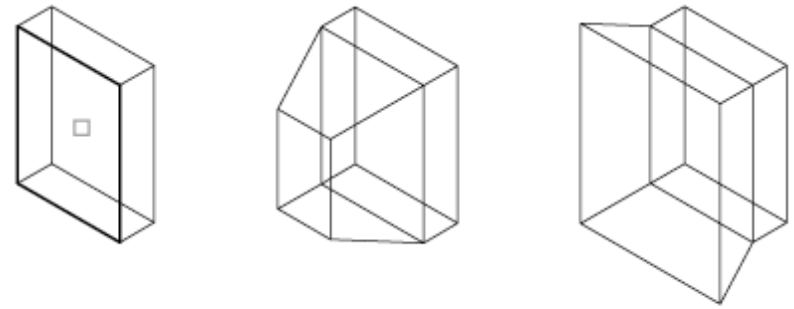

Specify height of extrusion or [Path]: *Specify a distance or enter* **p**



face selected face extruded

**Height of Extrusion** Sets the direction and height of the extrusion. Entering a positive value extrudes the face in the direction of its normal. Entering a negative value extrudes the face in the direction opposite to its normal.

Specify angle of taper for extrusion <0>: *Specify an angle between -90 and +90 degrees or press* ENTER

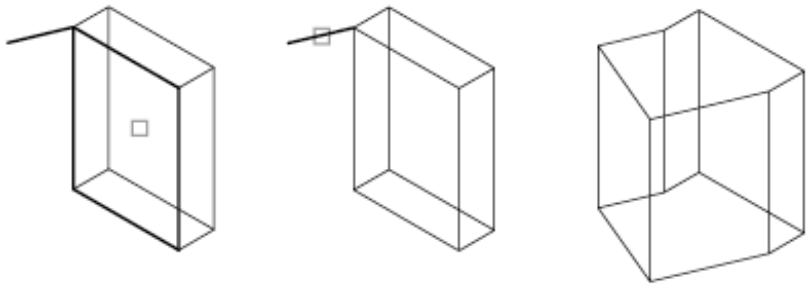


face selected positive angle extruded face negative angle extruded face

Tapering the selected face with a positive angle tapers the face in, and a negative angle tapers the face out. The default angle, 0, extrudes the face perpendicular to its plane. All selected faces in the selection set are tapered to the same value. If you specify a large taper angle or height, you can cause the face to taper to a point before it reaches the extrusion height.

**Path** Sets a path for the extrusion path based on a specified line or curve. All the profiles of the selected face are extruded along the chosen path to create the extrusion.

Select extrusion path: *Use an object selection method*




Lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines can be paths. The path should not lie on the same plane as the face, nor should it have areas of high curvature.

The extruded face starts from the plane of the profile and ends on a plane perpendicular to the path at the path's endpoint. One of the endpoints of the path should be on the plane of the profile; if not, the path is moved to the center of the profile.

If the path is a spline, the path should be perpendicular to the plane of the profile and at one of the endpoints of the path. If not, the profile is rotated to be perpendicular to the spline path. If one of the endpoints of the spline is on the plane of the face, the face is rotated about the point; otherwise, the spline path is moved to the center of the profile and the profiles are rotated about its center.

If the path contains segments that are not tangent, the object is extruded along each segment and then the joint along the plane is mitered, bisecting the angle formed by the segments. If the path is closed, the profile lies on the miter plane. This allows the start and end sections of the solid to match up. If the profile is not on the miter plane, the path is rotated until it is on the miter plane.

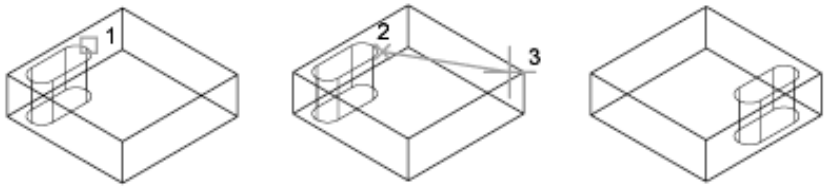
### Move

Moves the selected face on a 3D solid object to a specified height or distance. You can select multiple faces at one time.

Select faces or [Undo/Remove]: *Select one or more faces or enter an option*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed.

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1), enter an option, or press* ENTER
Specify a base point or displacement: *Specify a base point (2)*
Specify a second point of displacement: *Specify a point (3) or press* ENTER




face selected | base point and second point selected | face moved

The two points you specify define a displacement vector that indicates how far the selected face is moved and in what direction. SOLIDEDIT uses the first point as a base point and places a single copy relative to the base point. If you specify a single point, usually entered as a coordinate, and then press ENTER, the coordinate is used as the new location.

### Rotate

Rotates one or more faces or a collection of features on a solid about a specified axis.

Select faces or [Undo/Remove]: *Select one or more faces or enter an option*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed.

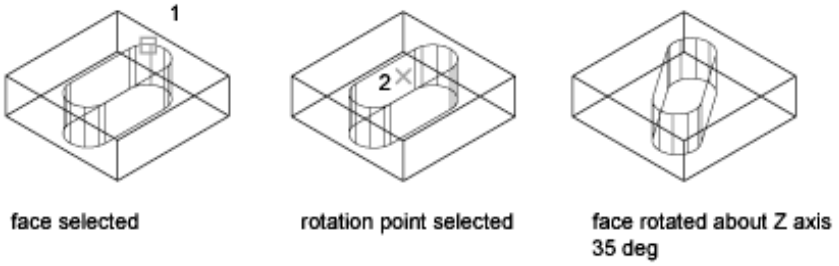
Select faces or [Undo/Remove/ALL]: *Select one or more faces, enter an option, or press* ENTER
Specify an axis point or [Axis by object/View/Xaxis/Yaxis/Zaxis] <2points>: *Enter an option, specify a point, or press* ENTER

### Axis Point, 2 Points

Use two points to define the axis of rotation. Pressing ENTER at the main Rotate prompt displays the following prompts. Specifying a point at the main prompt skips the prompt for the first point.

Specify the first point on the rotation axis: *Specify a point (1)*
Specify the second point on the rotation axis: *Specify a point (2)*

Specify a rotation angle or [Reference]: *Specify an angle or enter* **r**




**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference (starting) angle <0>: *Specify the starting angle*

Specify the ending angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### Axis by Object

Aligns the axis of rotation with an existing object. You can select the following objects:

- *Line:* Aligns the axis with the selected line.
- *Circle:* Aligns with the 3D axis of the circle (perpendicular to the plane of the circle and passing through the center of the circle).
- *Arc:* Aligns with the 3D axis of the arc (perpendicular to the plane of the arc and passing through the center of the arc).
- *Ellipse:* Aligns with the 3D axis of the ellipse (perpendicular to the plane of the ellipse and passing through the center of the ellipse).
- *2D polyline:* Aligns with the 3D axis formed by the polyline's start points and endpoints.
- *3D polyline:* Aligns with the 3D axis formed by the polyline's start points and endpoints.
- *Spline:* Aligns with the 3D axis formed by the spline's start points and endpoints.

Select a curve to be used for the axis: *Use an object selection method*
Specify a rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference (starting) angle <0>: *Specify the starting angle*

Specify the ending angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### View

Aligns the axis of rotation with the viewing direction of the current viewport that passes through the selected point.

Specify the origin of the rotation <0, 0, 0>: *Specify a point*
Specify a rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference (starting) angle <0>: *Specify the starting angle*

Specify the ending angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

### X Axis, Y Axis, Z Axis

Align the axis of rotation with the axis (*X*, *Y*, or *Z*) that passes through the selected point.

Specify the origin of the rotation <0, 0, 0>: *Specify a point*
Specify a rotation angle or [Reference]: *Specify an angle or enter* **r**
**Rotation Angle** Rotates the object about the selected axis the specified amount from the current orientation.

**Reference** Specifies the reference angle and the new angle.

Specify the reference (starting) angle <0>: *Specify the starting angle*

Specify the ending angle: *Specify the ending angle*

The difference between the starting angle and the ending angle is the computed rotation angle.

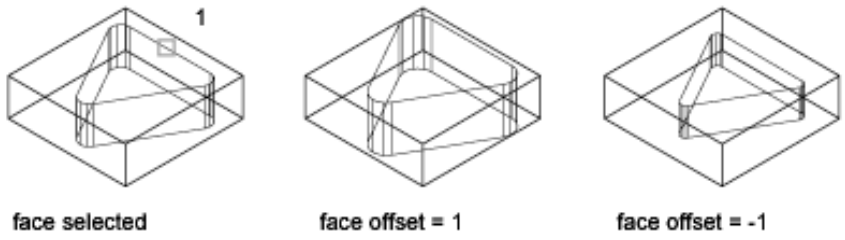
### Offset

Offsets faces equally by a specified distance or through a specified point. A positive value increases the size or volume of the solid, a negative value decreases the size or volume of the solid.

Select faces or [Undo/Remove]: *Select one or more faces or enter an option (1)*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces, enter an option, or press* ENTER
Specify the offset distance: *Specify a distance*




Specify a positive value to increase the size of the solid or a negative value to decrease the size of the solid.

**NOTE** Holes inside a solid object offset smaller as the volume of the solid gets larger.

### Taper

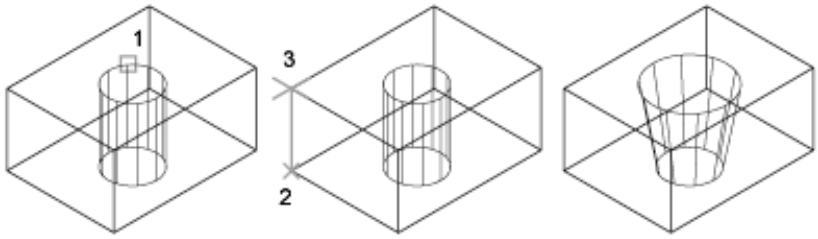
Tapers faces with an angle. The rotation of the taper angle is determined by the selection sequence of the base point and second point along the selected vector.

Select faces or [Undo/Remove]: *Select one or more faces or enter an option*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1), enter an option, or press* ENTER
Specify the base point: *Specify a base point (2)*
Specify another point along the axis of tapering: *Specify a point (3)*

Specify the taper angle: *Specify an angle between -90 and +90 degrees*




Tapering the selected face with a positive angle tapers the face in, and a negative angle tapers the face out. The default angle, 0, extrudes the face perpendicular to its plane. All selected faces in the selection set are tapered to the same value.

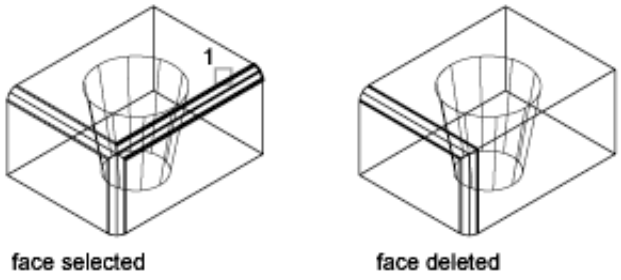
### Delete

Deletes or removes faces, including fillets and chamfers.

Select faces or [Undo/Remove]: *Select one or more faces*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1), enter an option, or press* ENTER
Solid validation started.
Solid validation completed.



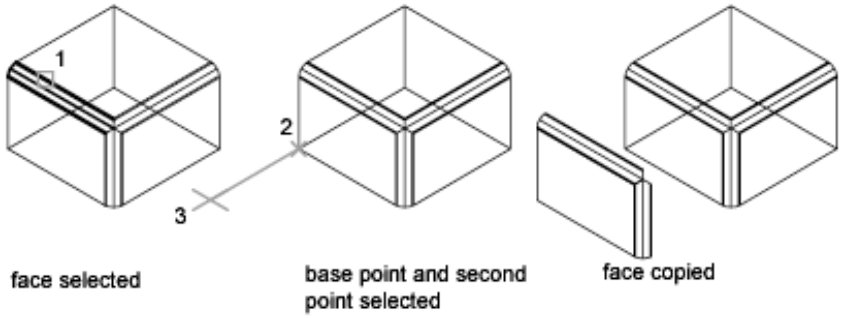
### Copy

Copies faces as a region or a body. If you specify two points, SOLIDEDIT uses the first point as a base point and places a single copy relative to the base point. If you specify a single point (usually entered as a coordinate) and then press ENTER, SOLIDEDIT uses the coordinate as the new location.

Select faces or [Undo/Remove]: *Select one or more faces*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces (1), enter an option, or press* ENTER
Specify a base point or displacement: *Specify a base point (2)*
Specify a second point of displacement: *Specify a point (3)*




### Color

Changes the color of faces.

Select faces or [Undo/Remove]: *Select one or more faces*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces, enter an option, or press* ENTER

The Select Color dialog box (page 251) is displayed.

### Material

Assigns a material to selected faces.

Select faces or [Undo/Remove]: *Select one or more faces*

The descriptions of the Undo, Remove and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: *Select one or more faces, enter an option, or press* ENTER
Enter new Material name <ByLayer>: *Enter the name of the material, or press* ENTER

---

**NOTE** The name of a material can be found by opening the Materials window and selecting the material swatch to display the name in the Name field.

---

### Undo

Reverses actions as far back as the beginning of the SOLIDEDIT session.

### Exit

Exits the face-editing options and displays the Enter a Solids Editing Option prompt.

### Edge

Edits 3D solid objects by changing the color of or copying individual edges.

Enter an edge editing option [Copy/coLor/Undo/eXit] <eXit>: *Enter an option or press* ENTER

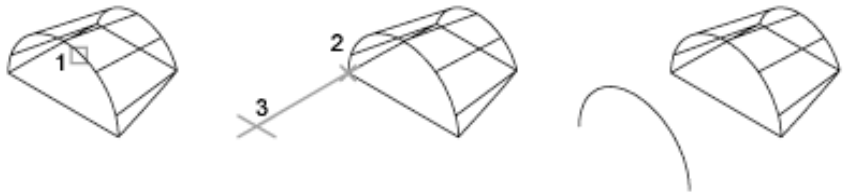
### Copy

Copies 3D edges. All 3D solid edges are copied as a line, arc, circle, ellipse, or spline.

Select edges or [Undo/Remove]: *Select one or more edges or enter an option*

After you select edges or enter an option, the following prompt is displayed:

Select edges or [Undo/Remove]: *Select one or more edges (1) or press* ENTER
Specify a base point of displacement: *Specify a base point (2)*
Specify a second point of displacement: *Specify a point (3)*




### Undo

Cancels selection of the edges you added most recently to the selection set. The previous prompt is displayed. If all edges have been removed, the following prompt is displayed:

Edge selection has been completely undone

### Remove

Removes previously selected edges from the selection set. The previous prompt is displayed.

Remove edges or [Undo/Add]: *Select one or more edges, enter an option, or press* ENTER

**Undo** Cancels selection of the edges you added most recently to the selection set. The previous prompt is displayed. If no edges are currently selected, the following prompt is displayed:

Edge selection has been completely undone

**Add** Adds edges to the selection set.

Select edges or [Undo/Remove]: *Select one or more edges or enter an option*

- *Undo:* Cancels selection of the edges you added most recently to the selection set. The previous prompt is displayed.
- *Remove:* Removes previously selected edges. The previous prompt is displayed.

### Color

Changes the color of edges.

Select edges or [Undo/Remove]: *Select one or more edges or enter an option*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Copy. After you select edges or enter an option, the Select Color dialog box (page 251) is displayed.

### Undo

Reverses actions as far back as the beginning of the SOLIDEDIT session.

### Exit

Exits the face-editing options and displays the Enter a Solids Editing Option prompt.

### Body

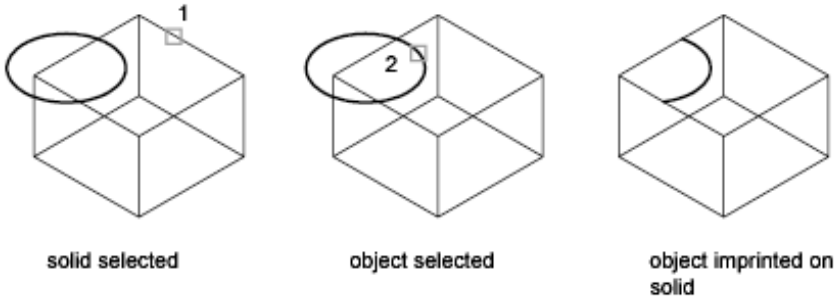
Edits the entire solid object by imprinting other geometry on the solid, separating the solid into individual solid objects, shelling, cleaning, or checking the selected solid.

Enter a body editing option [Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: *Enter an option or press* ENTER

### Imprint

Imprints an object on the selected solid. The object to be imprinted must intersect one or more faces on the selected solid in order for imprinting to be successful. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids.

Select a 3D solid: *Select an object (1)*
Select an object to imprint: *Select an object (2)*
Delete the source object <N>: *Enter* **y** *or press* ENTER
Select an object to imprint: *Select an object or press* ENTER



### Separate Solids

Separates 3D solid objects with disjointed volumes into independent 3D solid objects.

Select a 3D solid: *Select an object*

---

**NOTE** Separating solids does not separate Boolean objects that form a single volume.

---

### Shell

Shelling creates a hollow, thin wall with a specified thickness. You can specify a constant wall thickness for all the faces. You can also exclude faces from the shell by selecting them. A 3D solid can have only one shell. New faces are created by offsetting existing ones outside their original positions.

Select a 3D solid: *Select an object*
Remove faces or [Undo/Add]: *Select one or more faces or enter an option*

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

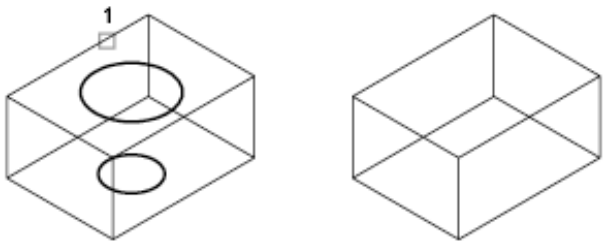
Remove faces or [Undo/Add/ALL]: *Select a face (1), enter an option, or press* ENTER
Enter the shell offset distance: *Specify a distance*


Specifying a positive value creates a shell to the inside perimeter of the solid; specifying a negative value creates a shell to the outside perimeter of the solid.

### Clean

Removes shared edges or vertices having the same surface or curve definition on either side of the edge or vertex. Removes all redundant edges, vertices, and unused geometry. Does not remove imprinted edges.

Select a 3D solid: *Select an object (1)*




solid selected

solid cleaned

### Check

Validates the 3D solid object as a valid solid, independent of the *SOLIDCHECK* setting.

Select a 3D solid: *Select an object*
This object is a valid ShapeManager solid.

### Undo

Undoes the editing action.

### Exit

Exits the face-editing options and displays the Enter a Solids Editing Option prompt.

### Undo

Undoes the editing action.

### Exit

Exits the SOLIDEDIT command.

# SOLPROF

## Quick Reference

Creates profile images of three-dimensional solids in paper space

Draw ➤ Modeling ➤ Setup ➤ ProfileAt the Command prompt, enter solprof.

**solprof**

Select objects: *Use an object selection method*

Display hidden profile lines on separate layer? [Yes (page 1330)/No (page 1330)] <Y>: *Enter* **y** *or* **n**, *or press* ENTER

**Yes** Generates only two blocks: one for the visible lines and one for the hidden lines of the entire selection set. When you generate hidden lines, solids can partially or completely hide other solids. The visible profile block is drawn in the BYLAYER linetype, and the hidden profile block is drawn in the HIDDEN linetype (if loaded). The visible and hidden profile blocks are placed on uniquely named layers using the following naming conventions:

PV-viewport handle for the visible profile layer

PH-viewport handle for the hidden profile layer

For example, if you create a profile in a viewport whose handle is 4B, the blocks containing the visible lines are inserted on layer PV-4B, and the block containing the hidden lines (if requested) is inserted on layer PH-4B. If these layers do not exist, the command creates them. If the layers do exist, the blocks are added to the information already on the layers.

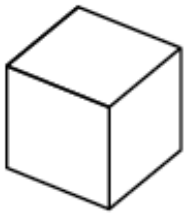
---

**NOTE** To determine the handle of a viewport, select the viewport while in paper space and use the *LIST* command. Choose a layout tab to move from model space to paper space.

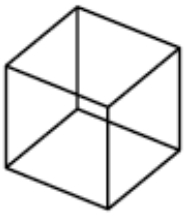
---

V does not change the display of layers; if you want to view only the profile lines you've created, turn off the layer containing the original solid (usually the current layer).

**No** Treats all profile lines as visible lines and creates a block for the profile lines of each selected solid. All profile lines for each solid in the selection set are generated, even if a solid is partially or completely obscured by another solid. The visible profile blocks are drawn in the same linetype as the original solid and placed on a uniquely named layer using the naming convention described under the Yes option.



profile with hidden lines removed



profile with hidden lines displayed

**NOTE** Solids that overlap each other (share some common volume) produce dangling edges if you request hidden-line removal. This happens because the edges must be broken at the point where they enter another solid to separate them into visible and hidden portions. You can eliminate dangling edges by combining the overlapping solids (using *UNION*) before generating a profile.

The next prompt determines whether 2D or 3D objects are used to represent the visible and hidden lines of the profile.

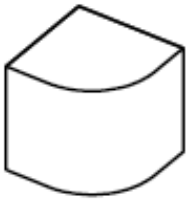
Project profile lines onto a plane? [Yes/No] <Y>: *Enter* **y** *or* **n**, *or press* ENTER

**Yes** Creates the profile lines with 2D objects. The 3D profile is projected onto a plane normal to the viewing direction and passing through the origin of the UCS. SOLPROF cleans up the 2D profile by eliminating lines that are parallel to the viewing direction and by converting arcs and circles that are viewed on edge into lines.

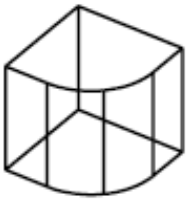
**No** Creates the profile lines with 3D objects.

The next prompt determines whether tangential edges are displayed. A tangential edge is the transition line between two tangent faces. It's the imaginary edge at which two faces meet and are tangent. For example, if you fillet the edge of a box, tangential edges are created where the cylindrical face of the fillet blends into the planar faces of the box. Tangential edges are not shown for most drafting applications.

Delete tangential edges? [Yes/No] <Y>: *Enter* **y** *or* **n**, *or press* ENTER



profile with tangential edges deleted



profile with tangential edges retained

# SOLVIEW

## Quick Reference

Creates layout viewports using orthographic projection to lay out multi- and sectional view drawings of 3D solids and body objects

Draw ➤ Modeling ➤ Setup ➤ ViewAt the Command prompt, enter solview.
**solview**

---

**NOTE** SOLVIEW must be run on a layout tab. If the Model tab is current, the last active layout tab is made current.

---

Enter an option [Ucs (page 1333)/Ortho (page 1335)/Auxiliary (page 1335)/Section (page 1336)]: *Enter an option or press* ENTER *to exit the command*

SOLVIEW places the viewport objects on the VPORTS layer, which it creates if it does not already exist. The view-specific information that is saved with each viewport you create is used by *SOLDRAW* to generate the final drawing view.

SOLVIEW creates layers that SOLDRAW uses to place the visible lines and hidden lines for each view, *view name*-VIS, *view name*-HID, *view name*-HAT, and a layer where you can place dimensions that are visible in individual viewports, *view name*-DIM.

---

**WARNING** The information stored on these layers is deleted and updated when you run *SOLDRAW*. Do not place permanent drawing information on these layers.

---

**NOTE** SOLVIEW is defined by the *acsolids.arx* application and is intended to be used only interactively.

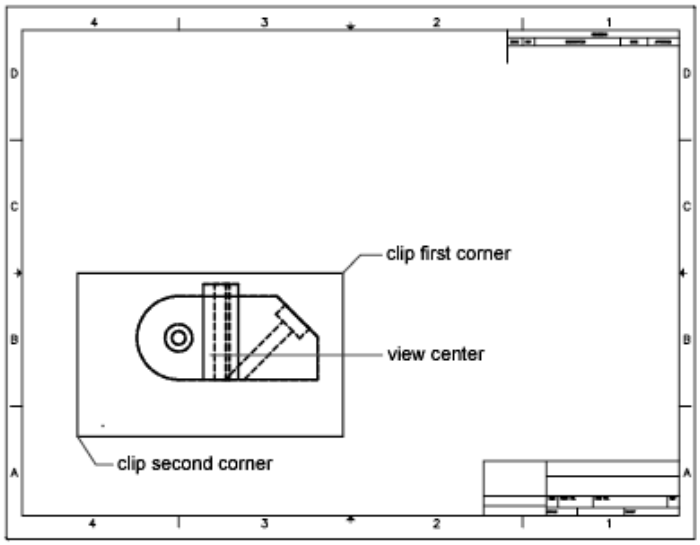
---

### UCS

Creates a profile view relative to a user coordinate system. If no viewports exist in your drawing, the UCS option is a good way to create an initial viewport from which other views can be created. All other SOLVIEW options require an existing viewport.

You have the option of using the current UCS or a previously saved one as the profile plane. The viewport projection is created parallel to the *XY* plane of the UCS with the *X* axis facing right and the *Y* axis upward.

Enter an option [Named/World/?/Current] <*current*>: *Enter an option or press* ENTER




**Named** Uses the *XY* plane of a named UCS to create a profile view.

Enter name of UCS to restore: *Enter the name of an existing UCS*

Enter view scale <1.0>: *Enter a number or press* ENTER

Enter the name of the UCS you want to use and the scale of your view. Entering a scale is equivalent to zooming your viewport by a factor relative to paper space. The default value is 1:1.

Specify view center: *Specify a point and press* ENTER *when done*

The center is based on the current model space extents. You can try several points until you are satisfied with the view's location.

At the next prompt, specify the opposite corners of the viewport.

Specify the first corner of viewport: *Specify a point*

Specify the other corner of viewport: *Specify a point*

Enter view name: *Enter a name for the view*

SOLVIEW returns to the initial prompt.

**World** Uses the *XY* plane of the WCS to create a profile view.

Enter view scale <1.0>: *Enter a number or press* ENTER

Enter the name of the UCS you want to use and the scale of your view. Entering a scale is equivalent to zooming your viewport by a factor relative to paper space. The default value is 1:1.

Specify view center: *Specify a point and press* ENTER *when done*

The center is based on the current model space extents. You can try several points until you are satisfied with the view's location.

At the next prompt, specify the opposite corners of the viewport.

Specify the first corner of viewport:

Specify the other corner of viewport:

Enter view name: *Enter a name for the view*

SOLVIEW returns to the initial prompt.

**?—List Named UCSs** Lists the names of existing user coordinate systems. The list is filtered using the wild-card combinations you enter (wild-card characters accepted by the UCS command are valid).

Enter UCS names to list <*>: *Enter a name or press* ENTER *to list all UCSs*

After the list is displayed, press any key to return to the first prompt.

**Current** Uses the *XY* plane of the current UCS to create a profile view.

Enter view scale <1.0>: *Enter a number or press* ENTER

Enter the name of the UCS you want to use and the scale of your view. Entering a scale is equivalent to zooming your viewport by a factor relative to paper space. The default value is 1:1.

Specify view center: *Specify a point and press* ENTER *when done*

The center is based on the current model space extents. You can try several points until you are satisfied with the view's location.

At the next prompt, specify the opposite corners of the viewport.

Specify the first corner of viewport:

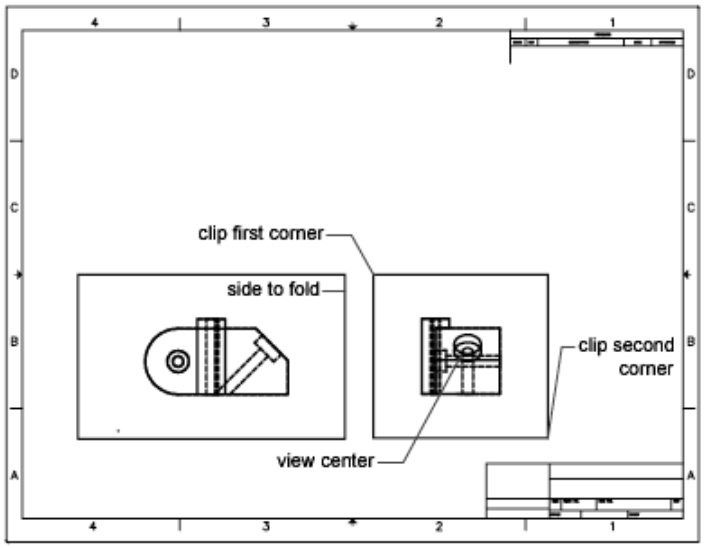
Specify the other corner of viewport:

Enter view name: *Enter a name for the view*

SOLVIEW returns to the initial prompt.

### Ortho

Creates a folded orthographic view from an existing view.




Once you select the side of the viewport you want to use for projecting the new view, a rubber-band line perpendicular to the side of the viewport helps you locate the center of the new view. You can try several points until you are satisfied with the view's location.

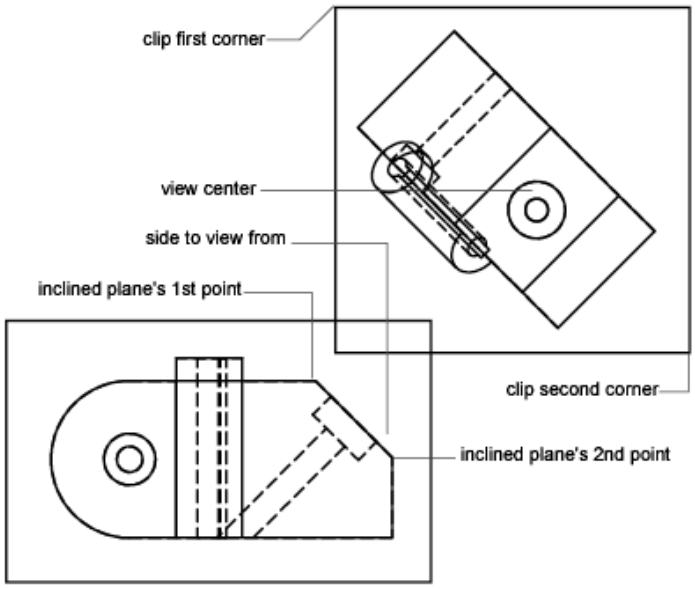
Specify side of viewport to project: *Select the edge of a viewport*
Specify view center: *Specify a point and press* ENTER

At the next prompt, specify the opposite corners of the viewport.

Specify first corner of viewport:
Specify opposite corner of viewport:
Enter view name: *Enter a name for the view*

### Auxiliary

Creates an auxiliary view from an existing view. An auxiliary view is one that is projected onto a plane perpendicular to one of the orthographic views and inclined in the adjacent view.




Two points define the inclined plane used for the auxiliary projection. Both points must be located in the same viewport.

Specify first point of inclined plane:
Specify second point of inclined plane:

The next point determines the side from which you view the plane.

Specify side to view from: *Specify a point*

A rubber-band line perpendicular to the inclined plane helps you select the center of the new viewport. You can try several points until you are satisfied with the view's location.

Specify view center: *Specify a point and press* ENTER

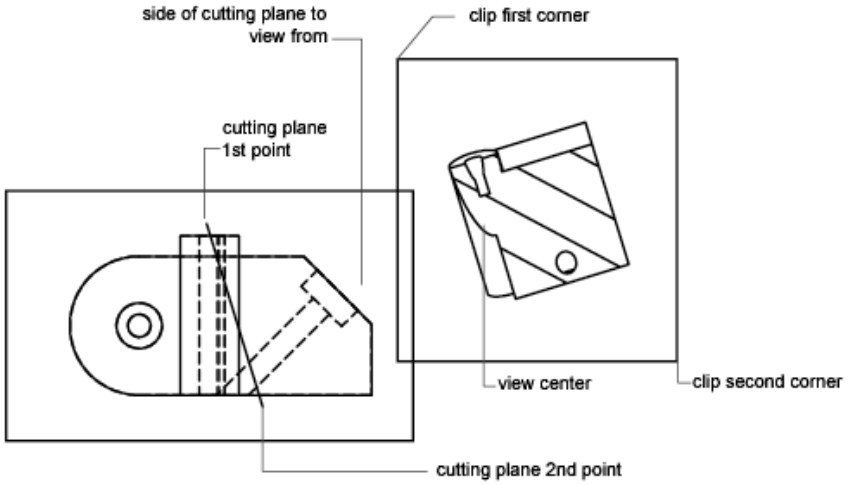
At the next prompt, specify the opposite corners of the viewport.

Specify first corner of viewport:
Specify opposite corner of viewport:
Enter view name: *Enter a name for the view*

### Section

Creates a drafting sectional view of solids, complete with cross-hatching. When you use *SOLDRAW* on a sectional view created with this option, it creates a temporary copy of the solids and uses *SLICE* to perform the operation at the

cutting plane that you define. SOLDRAW then generates a profile of the visible half of the solids and discards the original copy. Finally SOLDRAW sections the solids. Solids not crossing the cutting plane are generated as full profiles. Because drafting standards recommend not drawing hidden lines in sectional views, SOLVIEW freezes the *View Name*-HID layer.




In the original viewport, specify two points to define the sectioning plane.

Specify first point of cutting plane:
Specify second point of cutting plane:

Define the viewing side by specifying a point on one side of the cutting plane.

Specify side to view from: *Specify a point*

Enter the scale of the new view. Entering a scale is equivalent to zooming your viewport by a factor relative to paper space. The default value is a 1:1 scale, which is equivalent to **zoom 1.0xp**.

Enter view scale <*current*>: *Enter a positive number*

At the next prompt, specify the center of the new viewport. If you accepted the default scale (by pressing ENTER), a rubber-band line perpendicular to the sectioning plane helps you locate the center of the new view. Otherwise, you can place the view anywhere. You can try several points until you are satisfied with the view's location.

Specify view center: *Specify a point and press* ENTER

At the next prompt, specify the opposite corners of the viewport.

Specify first corner of viewport:
Specify opposite corner of viewport:
Enter view name: *Enter a name for the view*

# SPACETRANS

## Quick Reference

Calculates equivalent model space and paper space lengths in a layout

Text 
**spacetrans**

In a layout, when in model space, the prompt is displayed as follows:

Specify paper space distance <1.000>: *Enter a length in paper space to convert to the equivalent length in model space*

In a layout, when in paper space, the prompt is displayed as follows:

Select a viewport: *Pick a layout viewport object (this prompt is displayed when more than one viewport object is available in the layout)*
Specify model space distance <1.000>: *Enter a length in model space to convert to the equivalent length in paper space*

SPACETRANS converts lengths, typically text heights, from either model space or paper space to its equivalent length in the other space. It is intended to be invoked transparently at a prompt for text height or other length value. When used at the command prompt, SPACETRANS displays the computed length equivalent at the command prompt.

**NOTE** This command is not available from the Model tab or in a perspective view.

# SPELL

## Quick Reference

Checks spelling in a drawing


Text

Tools ➤ Spelling. At the Command prompt, enter spell.
**spell** (or '**spell** for transparent use)
Text, Check Spelling

When you enter **spell** at the command prompt, the Check Spelling dialog box (page 1339) is displayed. Select the Start button to begin the spelling check.

If Check Spelling is set to Entire Drawing, spelling is checked on model space, then on layouts 1 and 2 in paper space. Regardless of what current drawing area is showing, once Check Spelling starts, the drawing window changes to display the content that is currently being checked. If a flagged word is identified, the drawing area highlights and zooms to that word.

**NOTE** Invisible text such as text on hidden layers, and hidden block attributes is not checked. Non-uniformly scaled blocks and objects not on the supported annotation scale are also not checked.

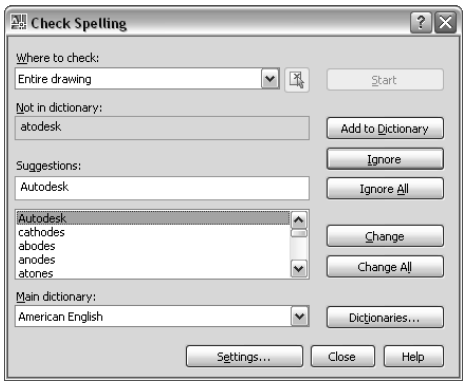
# Check Spelling Dialog Box

## Quick Reference

Tools ➤ Spelling
**spell** (or '**spell** for transparent use)

Corrects the spelling in text objects and all dimension text created with *TEXT*, *MTEXT*, *LEADER*, and *ATTDEF*.

In attributes, only the attribute values are checked. The spelling in text objects within block references and nested block references is checked, but spell checking in block definitions is performed only if the associated block reference has been selected.




**Where to Check** Displays the areas you want checked for spelling. Three options are available: Entire Drawing, Current Space/layout, and Selected Objects.

**Not in Dictionary** Displays the word identified as misspelled.

**Suggestions** Displays a list of suggested replacement words from the current dictionary. The first suggestion in the list box is highlighted in both Suggestions areas.

You can select another replacement word from the list, or edit or enter a replacement word in the top Suggestions text area.

**Main Dictionary** Lists the main dictionary options. The default dictionary will depend on the language setting.

**Start** Starts checking text for spelling errors.

**Ignore** Skips the current word.

**Ignore All** Skips all remaining words that match the current word.

**Add to Dictionary** Adds the current word to the current custom dictionary. The maximum word length is 63 characters.

**Change** Replaces the current word with the word in the Suggestions box.

**Change All** Replaces the current word in all selected text objects in the spell check area.

**Dictionaries** Displays the Dictionaries dialog box (page 1342).

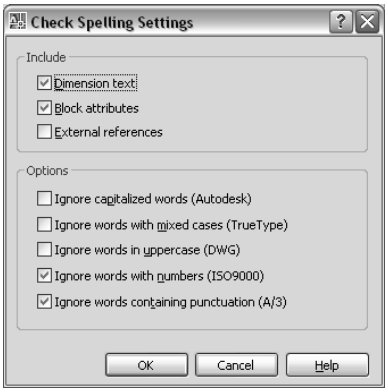
**Settings** Displays the Check Spelling Settings dialog box (page 1341).

# Check Spelling Settings Dialog Box

## Quick Reference

Tools ➤ SpellingAt the Command prompt, enter spell.
**spell** (or '**spell** for transparent use)

Specifies specific text options that will be checked in your drawing.




**Dimension Text** Searches dimension text. This is checked by default.

**Block Attributes** Searches block attributes text for spelling errors. This is selected by default.

**External References** Specifies to search external references for spelling errors.

**Ignore Capitalized Words** Specifies to ignore capitalized words.

**Ignore Words with Mix Cases** Specifies to ignore words that contain uppercase and lowercase letters.

**Ignore Words in Uppercase** Specifies to ignore words that are in all uppercase.

**Ignore Words with Numbers** Specifies to ignore words that include numbers.

**Ignore Words Containing Punctuation** Specifies to ignore words that contain punctuation.

# Dictionaries Dialog Box

## Quick Reference

Tools ➤ SpellingAt the Command prompt, enter spell.
**spell** (or '**spell** for transparent use)

Manages dictionaries. During a spelling check, the words in the drawing are matched to the words in the current main and current custom dictionaries. Any spelling exceptions that you identify with the Add option are stored in the custom dictionary you are currently using.

If you want to check spelling in another language, you can change to a different main dictionary. You can also create any number of custom dictionaries and switch between them as needed.


### Main Dictionary

Displays a list of language-specific dictionaries from which you can choose a different main dictionary. This dictionary is used in conjunction with the custom dictionary.

### Custom Dictionary

Displays the name of the current custom dictionary. The *.cus* extension is used for an AutoCAD custom dictionary.

**Current Custom Dictionary** Displays the list from which you can select a custom dictionary file to use. The selected dictionary file is used until another

file is selected. The list also includes a Manage custom dictionaries selection which brings the Manage Custom Dictionaries dialog box (page 1343).

**Content** Displays a list of the words that currently exist in the specified custom dictionary.

You can add words to or delete words from this list.

**Add** Adds the word that you enter in the box to the current custom dictionary. The maximum length is 63 characters.

**Delete** Deletes a word from current custom dictionary content.

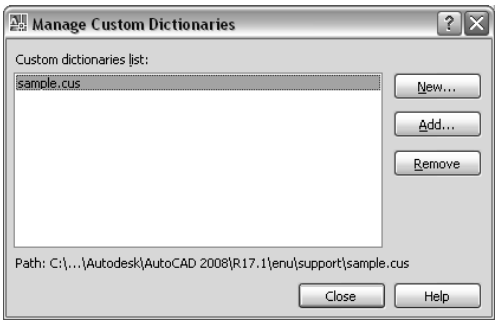
**Import** Imports words from another dictionary or word list into your current custom dictionary.

# Manage Custom Dictionaries Dialog Box

## Quick Reference

Tools ➤ SpellingAt the Command prompt, enter spell.
**spell** (or '**spell** for transparent use)

Manages your custom dictionaries allowing you to add or remove a dictionary.




**Custom Dictionaries List** Displays the custom dictionaries list. Only one dictionary can be selected at a time.

**New** Allows you to create a custom dictionary. The new dictionary is highlighted as the current one.

**NOTE** The filename for a custom dictionary cannot use any non-current code page characters in its name. If you are sharing a custom dictionary between different locals or languages do not use non-ASCII characters.

**Add** Allows you to add an existing custom dictionary from your file lists.

**Remove** Allows you to delete a custom dictionary file from your list.

# SPHERE

## Quick Reference

Creates a 3D solid sphere

Modeling 
Draw ➤ Modeling ➤ SphereAt the Command prompt, enter sphere.
**sphere**
3D Make panel, Sphere
Specify center point (page 1344) or [3P (page 1345)/2P (page 1345)/TTR (page 1345)]:
*Specify a point or enter an option*

### Center Point

Specifies the center point of the sphere.

When you specify the center point, the sphere is positioned so that its central axis is parallel to the *Z* axis of the current user coordinate system (UCS). Latitudinal lines are parallel to the *XY* plane.

Specify radius or [Diameter]: *Specify a distance or enter* **d**

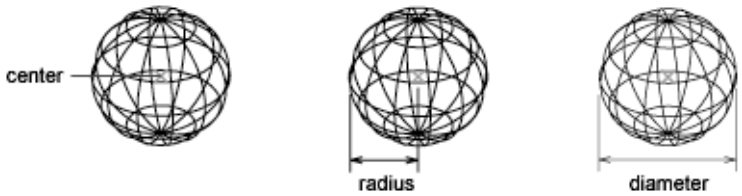
### Radius

Defines the radius of the sphere.

### Diameter

Defines the diameter of the sphere.

Specify diameter: *Specify a distance*




### 3P (Three Points)

Defines the circumference of the sphere by specifying three points anywhere in 3D space. The three specified points also define the plane of the circumference.

Specify first point: *Specify a point (1)*
Specify second point: *Specify a point (2)*
Specify third point: *Specify a point (3)*

### 2P (Two Points)

Defines the circumference of the sphere by specifying two points anywhere in 3D space. The plane of the circumference is defined by the *Z* value of the first point.

Specify first endpoint of diameter: *Specify a point (1)*
Specify second endpoint of diameter: *Specify a point (2)*

### TTR (Tangent, Tangent, Radius)

Defines the sphere with a specified radius tangent to two objects. The specified tangency points are projected onto the current UCS.

Specify point on object for first tangent: *Select a point on an object*
Specify point on object for second tangent: *Select a point on an object*
Specify radius <default>: *Specify a radius or press* ENTER *to specify the default radius value*

Initially, the default radius is not set to any value. During a drawing session, the default value for the radius is always the previously entered radius value for any solid primitive.

# SPLINE

## Quick Reference

Fits a smooth curve to a sequence of points within a specified tolerance

Draw 

Draw ➤ SplineAt the Command prompt, enter spline.

**spline**

Specify first point (page 1346) or [Object (page 1348)]: *Specify a point or enter* **o**

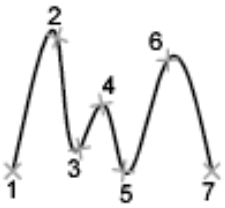
### First Point

Creates a spline using points you specify, using NURBS (nonuniform rational B-splines) mathematics.

Specify next point: *Specify a point*

Enter points until you have defined the spline curve. After you enter two points, the following prompt is displayed:

Specify next point or [Close/Fit Tolerance] <Start tangent>: *Specify a point, enter an option, or press* ENTER

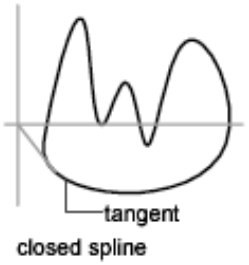



**Next Point** Continuing to enter points adds additional spline curve segments until you press ENTER. Enter **undo** to remove the last point specified. Once you press ENTER, you are prompted to specify the start tangent for the spline curve.

**Close** Closes the spline curve by defining the last point as co-incident with the first and making it tangent to the joint.

Specify tangent: *Specify a point or press* ENTER

Specify a point to define the tangent vector or use the Tangent and Perpendicular object snap modes to make the spline tangent or perpendicular to existing objects.




**Fit Tolerance** Changes the tolerance for fitting of the current spline curve. The spline curve is redefined so that it fits through the existing points according to the new tolerance. You can repeatedly change the fit tolerance, but doing so changes the fit tolerance for all the control points regardless of the control point that is selected.

Specify fit tolerance <*current*>: *Enter a value or press* ENTER

If you set the tolerance to 0, the spline curve passes through the fit points. Entering a tolerance greater than 0 allows the spline curve to pass through the fit points within the specified tolerance.

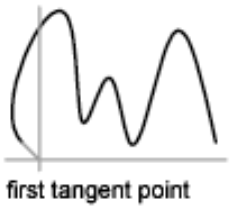
SPLINE returns to the previous prompt.


**Start Tangent** Defines the tangency for the first and last points of the spline curve.

Specify start tangent: *Specify a point or press* ENTER

The Specify Start Tangent prompt specifies the tangency of the spline curve at the first point.

Specify end tangent:Specify a point or press ENTER



The Specify End Tangent prompt specifies the tangency of the spline curve at the last point.


If you specify tangency at both endpoints of the spline, you can enter a point or use the Tangent and Perpendicular object snap modes to make the spline tangent or perpendicular to existing objects. Press ENTER to calculate default tangents.

#### Object

Converts 2D or 3D quadratic or cubic spline-fit polylines to equivalent splines and (depending on the setting of the *DELOBJ* system variable) deletes the polylines.

Select objects to convert to splines...
Select objects: *Select 2D or 3D spline-fit polylines and press* ENTER *when you finish*

# SPLINEDIT

## Quick Reference

Edits a spline or spline-fit polyline

Modify II 
Modify ➤ Object ➤ SplineAt the Command prompt, enter splinedit.
Select a spline to edit, right-click in the drawing area, and choose Spline Edit.
**splinedit**
Select spline:
Enter an option [Fit data (page 1349)/Close (page 1352)/Move vertex (page 1353)/Refine (page 1353)/rEverse (page 1355)/Undo (page 1355)]:

When you select a spline object or spline-fit polyline, grips appear at the control points.

If the selected spline is closed, the Close option changes to Open. If the selected spline has no fit data, the Fit Data option is not available. Fit data consists of all fit points, the fit tolerance, and tangents associated with splines created with the *SPLINE* command.

A spline can lose its fit data if you

- Use the Purge option while editing fit data
- Refine the spline by elevating the order, adding a control point, or changing the weight of a control point
- Change the fit tolerance
- Move a control point
- Trim, break, stretch, or lengthen the spline

**NOTE** SPLINEDIT automatically converts splined polylines to spline objects. A splined polyline is converted even if you select it and immediately exit SPLINEDIT.

### Fit Data

Edits fit data using the following options:

Enter a fit data option
[Add/Close/Delete/Move/Purge/Tangents/toLerance/eXit] <eXit>: *Enter an option or press* ENTER

**NOTE** If the selected spline is closed, the Close option is replaced by the Open option.

### Add

Adds fit points to a spline.

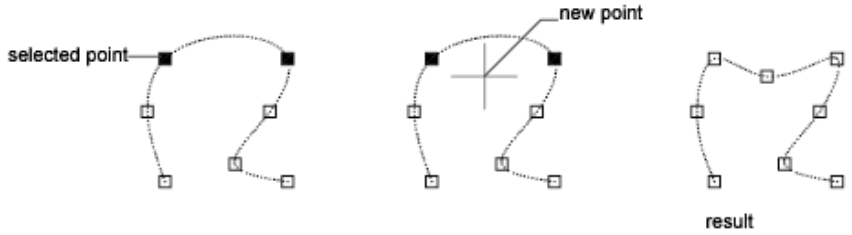
Specify control point <exit>: *Specify a control point or press* ENTER
Specify new point <exit>: *Specify a point or press* ENTER
Specify new point <exit>: *Specify a point or press* ENTER

When you select a point, SPLINEDIT highlights it and the next point and interprets the new point to go between the highlighted points. Use the Undo option to remove the last point added. Selecting the last point on an open spline highlights only that point, and SPLINEDIT adds the new point after

the last. If you select the first point on an open spline, you have the option of placing the new point before or after the first point.

Specify new point or [After/Before] <exit>: *Specify a point, enter an option, or press* ENTER
Specify new point <exit>: *Specify a point or press* ENTER

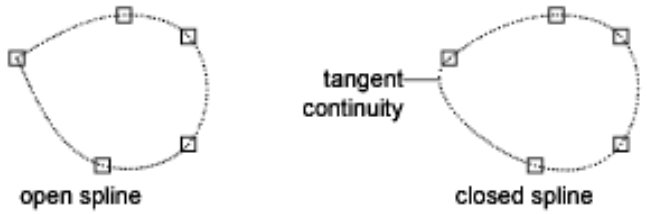
The point is added, and the spline is refitted through the new set of points.




### Close/Open

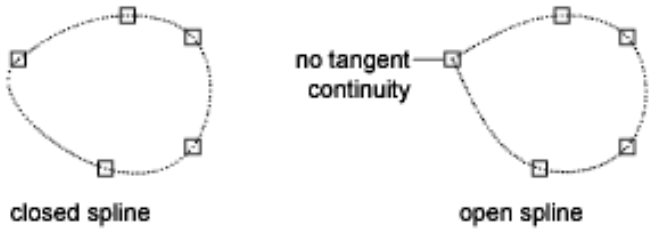
If the selected spline is closed, the Close option is replaced by the Open option.

**Close** Closes an open spline and makes it tangent continuous (smooth) at its endpoints. If the spline's start point and endpoint are the same, this option makes the spline tangent continuous at both points.




**Open** Opens a closed spline. If the spline's start point and endpoint were the same before you used the Close option to make it tangent continuous at both points, the Open option returns the spline to its original state. The start point and endpoint remain the same but lose their tangent continuity (smoothness).

If the spline was open (its start point and endpoint were not the same) before you used the Close option to make it tangent continuous where the start point and endpoint meet, this option returns the spline to its original open state and removes tangent continuity.




### Delete

Removes fit points from a spline and refits the spline through the remaining points.

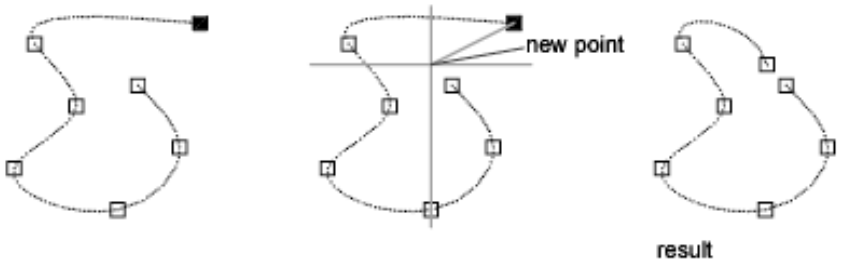
Specify control point <exit>: *Specify a control point or press* ENTER

### Move

Moves fit points to a new location.

Specify new location or [Next/Previous/Select Point/eXit] <N>: *Specify a point, enter an option, or press* ENTER

**New Location** Moves the selected point to the new location you specify. The previous prompt is repeated.




**Next** Moves selection to the next point.

**Previous** Moves selection to the previous point.

**Select Point** Selects a point from the set of fit points.

Specify fit point <exit>: *Specify a fit point or press* ENTER

**Exit** Returns to the Enter a Fit Data Option prompt.

### Purge

Removes a spline's fit data from the drawing database. After purging the spline's fit data, the main SPLINEDIT prompt is displayed without the Fit Data option.

### Tangents

Edits the start and end tangents of a spline.

Specify start tangent or [System default]: *Specify a point, enter an option, or press* ENTER
Specify end tangent or [System default]: *Specify a point, enter an option, or press* ENTER

If the spline is closed, the prompt becomes Specify Tangent or [System Default].

The System Default option calculates the default tangents at the ends.

You can specify a point or use the Tangent or Perpendicular object snap mode to make the spline tangential or perpendicular to existing objects.

### Tolerance

Refits the spline to the existing points with new tolerance values.

Enter fit tolerance <*current*>: *Enter a value or press* ENTER


### Exit

Returns to the main SPLINEDIT prompt.

### Close/Open

If the selected spline is closed, the Close option changes to Open.

**Close** Closes an open spline and makes it tangent continuous (smooth) at its endpoints. If the spline's start point and endpoint are the same, this option makes the spline tangent continuous at both points.

**Open** Opens a closed spline. If the spline's start point and endpoint were the same before you used the Close option to make it tangent continuous at both points, the Open option returns the spline to its original state. The start point and endpoint remain the same but lose their tangent continuity (smoothness).

If the spline was open (its start point and endpoint were not the same) before you used the Close option to make it tangent continuous where the start point and endpoint meet, this option returns the spline to its original open state and removes tangent continuity.

### Move Vertex

Relocates a spline's control vertices and purges the fit points.

Specify new location or [Next/Previous/Select Point/eXit] <N>: *Specify a point, enter an option, or press* ENTER
**New Location** Moves the selected point to the new location you specify.

After you specify a new location for the selected point, SPLINEDIT recalculates and displays the spline with a new set of control points.

**Next** Moves selection to the next point. The point marker does not wrap around from the end to the start of the spline, even if the spline is closed.

**Previous** Moves selection to the previous point. The point marker does not wrap around from the start to the end of the spline, even if the spline is closed.

**Select Point** Selects a point from the set of control points.

Specify fit point <exit>: *Specify a point or press* ENTER

**Exit** Returns to the main SPLINEDIT prompt.

### Refine

Fine-tunes a spline definition.

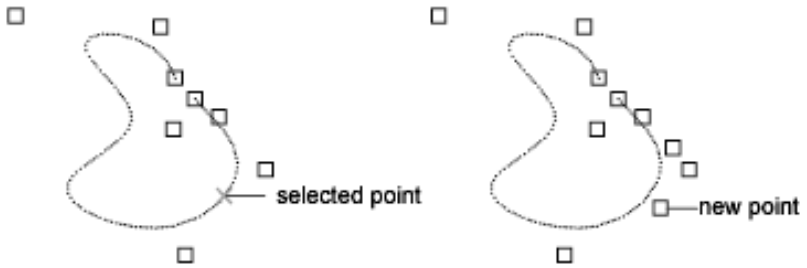
Enter a refine option [Add control point/Elevate order/Weight/eXit] <eXit>: *Enter an option or press* ENTER

### Add Control Point

Increases the number of control points that control a portion of a spline.

Specify a point on the spline <exit>: *Specify a point or press* ENTER

SPLINEDIT adds a new control point close to the point you select, between the two control points affecting that portion of the spline.




### Elevate Order

Increases the number of control points on the spline.

Enter new order <*current*>: *Enter an integer or press* ENTER

Entering a value greater than the current value increases the number of control points uniformly across the spline for more localized control. The maximum value for order is 26.

### Weight

Changes the weight at various spline control points. A larger weight value pulls the spline closer to the control point.

Enter new weight (current = *current*) or [Next/Previous/Select point/eXit] <N>: *Enter a value, enter an option, or press* ENTER

**New Weight** Recalculates the spline based on the new weight value at the selected control point. The larger the integer value, the more the spline is pulled toward the control point.

**Next** Moves selection to the next point. The point marker does not wrap around from the end to the start of the spline, even if the spline is closed.

**Previous** Moves selection to the previous point. The point marker does not wrap around from the start to the end of the spline, even if the spline is closed.

**Select Point** Selects a point from the set of control points.

Specify fit point <exit>: *Specify a point or press* ENTER

**Exit** Returns to the main Refine prompt.

### Exit

Returns to the main SPLINEDIT prompt.

#### Reverse

Reverses the spline's direction. This option is intended primarily for third-party applications.

#### Undo

Cancels the last editing action.

# SPOTLIGHT

## Quick Reference

Creates a spotlight

Lights 

View ➤ Render ➤ Light ➤ New SpotlightAt the Command prompt, enter spotlight.

**spotlight**

Light panel (click to extend), Create a Spot Light

Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*

Specify target location <1,1,1>: *Enter coordinate values or use the pointing device*

If the LIGHTINGUNITS system variable is set to 0, the following prompt is displayed:

Enter an option to change [Name (page 1356)/Intensity (page 1356)/Status (page 1356)/Hotspot (page 1356)/Falloff (page 1356)/shadoW (page 1357)/Attenuation (page 1358)/Color (page 1359)/eXit (page 1359)] <eXit>:

If the LIGHTINGUNITS system variable is set to 1 or 2, the following prompt is displayed:

Enter an option to change [Name (page 1356)/Intensity factor (page 1356)/Photometry (page 1356)/Status (page 1356)/Hotspot (page 1356)/Falloff (page 1356)/shadoW (page 1357)/filterColor (page 1359)/eXit (page 1359)] <eXit>:

---

**NOTE** When the LIGHTINGUNITS system variable is set to 1 or 2, the Attenuation option has no affect on the creation of the light. It is only maintained for scripting compatibility.

---

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity/Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00 - max float) <1.0000>:

### Hotspot

Specifies the angle that defines the brightest cone of light, which is known to lighting designers as the beam angle. This value can range from 0 to 160 degrees or the equivalent values based on *AUNITS* and *AUNITS*.

Enter hotspot angle (0.00-160.00) <45.0000>:

### Falloff

Specifies the angle that defines the full cone of light, which is also known as the field angle. This value can range from 0 to 160 degrees. The default is 50 degrees or the equivalent values based on *AUNITS* and *AUNITS*. The falloff angle must be greater than or equal to the hotspot angle.

Enter falloff angle (0.00-160.00) <50>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf] <On>:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived

power per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i**, you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*> :

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command option.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>:

**Off** Turns off display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory to use to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness to use to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Disk/Rect] <Disk>:

Specify the sample size by entering **a.**

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v.**

Enter Shape Visibility [Yes/No]<No>:

### Attenuation

Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit]<eXit>:
**Attenuation Type** Controls how light diminishes over distance. The farther away an object is from a spotlight, the darker the object appears. Attenuation is also known as decay.

Enter attenuation type [None/Inverse linear/inverse Squared] <Inverse linear>:

- None. Sets no attenuation. Objects far from the spotlight are as bright as objects close to the light.
- Inverse Linear. Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the spotlight; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.
- Inverse Squared. Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one

quarter as strong as at the spotlight; at a distance of 4 units, light is one sixteenth as strong.

**Use Limits** Specifies whether to use limits or not.

Limits [oN/oFf] <Off>:

**Attenuation Start Limit** Specifies the point where light starts as an offset from the center of the light. The default is 0.

Specify start limit offset <1.0000>:

---

**NOTE** Attenuation start limits and end limits are not supported by the OpenGL driver (*wopengl9.hdi*), but the Direct 3D driver (*direct3d9.hdi*) does support end limits for attenuation but does not support start limits. To identify your driver, enter **3dconfig**, and click Manual Tune. Look at the selected Driver Name in the Manual Performance Tuning dialog box.

---

**Attenuation End Limit** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point. Setting an end limit increases performance where the effect of lighting is so minimal that the calculations are wasted processing time.

Specify end limit offset <10.0000>:

### Color/Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]<255,255,255>:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L) <0,0,100>:

**Color Book** Specifies a color from a color book.

Enter Color Book name:

### Exit

Exits the command.

# STANDARDS

## Quick Reference

Manages the association of standards files with drawings.

CAD Standards
Tools ➤ CAD Standards ➤ ConfigureAt the Command prompt, enter standards.
**standards**

The Configure Standards dialog box (page 1360) is displayed.

# Configure Standards Dialog Box

## Quick Reference

CAD Standards
Tools ➤ CAD Standards ➤ ConfigureAt the Command prompt, enter standards.
**standards**

Associates the current drawing with a standards file and lists plug-ins used to check standards.

**Check Standards** Opens the Check Standards dialog box (page 243).

**Settings** Displays the CAD Standards Settings dialog box (page 1362), which specifies additional settings for the Configure Standards and Check Standards dialog boxes.

The Configure Standards dialog box contains the following tabs:

- Standards (page 1361)
- Plug-ins (page 1362)

### Standards Tab

Displays information about the standards files that are associated with the current drawing.

### Standards Files Associated with the Current Drawing

Lists all standards (DWS) files that are associated with the current drawing. To add a standards file, click Add Standards File. To remove a standards file, click Remove Standards File. If conflicts arise between multiple standards in this list (for example, if two standards specify layers of the same name but with different properties), the standards file that is shown first in the list takes precedence. To change the position of a standards file in the list, select it and click Move Up or Move Down.

You can use the shortcut menu to add, remove, or reorder files.

**Add Standards File** Associates a standards (DWS) file with the current drawing.

**Remove Standards File** Removes a standards file from the list. (Removing a standards file does not delete it but simply dissociates it from the current drawing.)

**Move Up** Moves a standards file up one position in the list.

**Move Down** Moves a standards file down one position in the list.

**Description** Provides summary information about the standards file currently selected in the list. You can add comments and a title to the description by opening the DWS file and using the DWGPROPS command. In the Drawing Properties dialog box, click the Summary tab.

### Plug-ins Tab

Lists and describes the standards plug-ins that are installed on the current system. A standards plug-in is installed for each of the named objects for which standards can be defined (layers, dimension styles, linetypes, and text styles). In the future, it is expected that third-party applications will be able to install additional plug-ins.

**Plug-ins Used When Checking Standards** Lists the standards plug-ins on the current system. You can specify which plug-ins to use when auditing a drawing by selecting plug-ins from this list.

**Description** Provides summary information about the standards plug-in currently selected in the list.

## CAD Standards Settings Dialog Box

### Quick Reference

CAD Standards

**Tools menu:** CAD Standards ➤ Configure

**standards** or **checkstandards**

Specifies additional settings for the Configure Standards dialog box (page 1360)and the Check Standards dialog box (page 243).

**Notification Settings** Sets an option for notification about standards violations.

**Disable Standards Notifications** Turns off notifications for standards violations and for missing standards files.

**Display Alert upon Standards Violation** Turns on notification for standards violations in the current drawing. An alert is displayed when you commit a standards violation. The alert informs you of how many nonstandard objects you created or edited while making changes to the drawing. Once the alert is displayed, you can choose to fix or not fix the standards violation. For drawings that use scripts and LISP routines, the alert is not displayed until the script or routine has been completed. (*STANDARDSVIOLATION* system variable)

**Display Standards Status Bar Icon** Displays an icon in the status bar when you open a file associated with a standards file and when you create or modify non-standard objects. (STANDARDSVIOLATION system variable)

**Check Standards Settings** Sets options for fixing violations and ignoring flagged problems.

**Automatically Fix Non-standard Properties** Sw itches between automatically fixing or not fixing nonstandard objects if a recommended fix is available. Recommended fixes are available only when a nonstandard object has a name that matches a standard object, but different properties. In this case, the properties of the standard object are applied to the nonstandard object. The Checking Complete alert summarizes the number of automatically fixed violations that occur after an audit.

This option is set for each system and applies to all drawings checked on a system.

**Show Ignored Problems** Switches between displaying or not displaying problems that have been flagged as ignored. If this option is checked, standards violations that have been flagged as ignored are displayed when an audit is performed on the current drawing.

**Preferred Standards File to Use for Replacements** Provides a list of standards files that control the default selection in the Replace With list in the Check Standards dialog box. Regardless of this setting, if a recommended replacement is found (this is the replacement preceded by a check mark), it is always the default selection. If no recommended replacement is found when a standards file (DWS) is selected in the CAD Standards Settings dialog box, the default selection in the Replace With list is the first replacement item from the selected standards file. If this setting is set to None, and no recommended replacement is found, no items will be selected in the Replace With list.

# STATUS

## Quick Reference

Displays drawing statistics, modes, and extents

Tools ➤ Inquiry ➤ StatusAt the Command prompt, enter status.
**status** (or '**status** for transparent use)

All coordinates and distances are displayed by STATUS in the format specified by *UNITS*.

STATUS reports the number of objects in the current drawing. This includes graphical objects such as arcs and polylines, and nongraphical objects such as layers and linetypes, and block definitions. When used at the DIM prompt,

STATUS reports the values and descriptions of all dimensioning system variables.

In addition, STATUS displays the following information.

**Model or Paper Space Limits Are** Displays the grid limits defined by *LIMITS*. The first line shows the *XY* coordinate of the limit's lower-left corner, stored in the *LIMMIN* system variable. The second line shows the *XY* coordinate of the limit's upper-right corner, stored in the *LIMMAX* system variable. The notation Off to the right of the *Y* coordinate value indicates that limits checking is set to 0.

**Model or Paper Space Uses** Displays the drawing extents, which includes all objects in the database and can exceed the grid limits. The first line shows the *XY* coordinate of the lower-left corner of the extents. The second line shows the *XY* coordinate of the upper-right corner. The notation Over to the right of the *Y* coordinate value indicates that the drawing extends outside the grid limits.

**Display Shows** Lists the portion of the drawing extents visible in the current viewport. The first line shows the *XY* coordinate of the display's lower-left corner. The second line shows the *XY* coordinate of the upper-right corner.

**Insertion Base Is** Displays the insertion point of the drawing, stored in the *INSBASE* system variable and expressed as an *X*,*Y*,*Z* coordinate.

**Snap Resolution Is** Displays the snap spacing in the *X* and *Y* directions, stored in the *SNAPUNIT* system variable.

**Grid Spacing Is** Displays the grid spacing in the *X* and *Y* directions, stored in the *GRIDUNIT* system variable.

**Current Space** Shows whether model space or paper space is active.

**Current Layout** Displays “Model” or the name of the current layout.

**Current Layer** Lists the current layer of the drawing, stored by the *CLAYER* system variable.

**Current Color** Lists the current color, stored by the *CECOLOR* system variable.

**Current Linetype** Lists the current linetype, stored by the *CELTYPE* system variable.

**Current Material** Lists the current material, stored by the *CMATERIAL* system variable.

**Current Lineweight** Lists the current lineweight, stored by the *CELWEIGHT* system variable.

**Current Plot Style** Lists the current plot style, stored by the *CPLOTSTYLE* system variable.

**Current Elevation** Lists the current elevation of the drawing, stored in the *ELEVATION* system variable.

**Thickness** Lists the current thickness, stored in the *THICKNESS* system variable.

**Fill, Grid, Ortho, Qtext, Snap, Tablet** Shows whether these modes are on or off.

**Object Snap Modes** Lists the running object snap modes specified by *OSNAP*.

**Free Dwg Disk Space** Lists the amount of disk space available on the drive specified for this program's temporary files.

**Free Temp Disk Space** Lists the amount of disk space available on the drive specified for temporary files.

**Free Physical Memory** Lists the amount of installed memory free on your system.

**Free Swap File Space** Lists the amount of free space in your swap file.

# STLOUT

## Quick Reference

Stores a solid in an ASCII or binary file

**stlout**
Select a single solid for STL output...
Select objects: *Use an object selection method and press* ENTER *when you finish*
Create a binary STL file? [Yes/No] <Yes>: *Enter* **y** *or press* ENTER *to create a binary file, or ente***n***create an ASCII file*

The Create STL File dialog box (a standard file selection dialog box (page 931)) is displayed.

The file is created with the *.stl* file name extension. The STL file format is compatible with stereolithography apparatus (SLA). The solid data is transferred to the SLA as a faceted representation of the model. The facets consist of a set of triangles (with outward pointing normals) that approximate the faces of the model. From the faceted data, the SLA workstation produces a set of contours that defines a series of layers representing the part to be built.

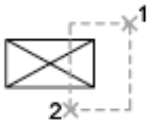
The *FACETRES* system variable determines how the solid is triangulated. Setting FACETRES to a higher value creates a finer mesh that more accurately represents the model and also creates a much larger file. The model must lie in the positive *XYZ* octant of the WCS.

# STRETCH

## Quick Reference

Moves or stretches the objects

Modify 
Modify ➤ StretchDoes not exist in the menus.
**stretch**
Select objects to stretch by crossing-window or crossing-polygon...
Select objects: *Use the cpolygon option or the crossing object selection method, and press* ENTER. *Individually selected objects and objects that are completely enclosed by the crossing selection are moved rather than stretched.*



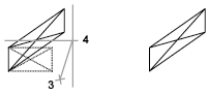

STRETCH moves only the vertices and endpoints that lie inside the crossing selection, leaving those outside unchanged. STRETCH does not modify 3D solids, polyline width, tangent, or curve-fitting information.

### Base Point

Specify base point or [Displacement] <*last displacement*>: *Specify a base point or enter displacement coordinates*
Specify second point or <*use first point as displacement*>: *Specify a second point, or press* ENTER *to use the previous coordinates as a displacement*

### Displacement

Specify displacement <*last value*>: *Enter displacement values for X,Y (and optionally Z)*




If you enter a second point, the objects are stretched the vector distance from the base point to the second point. If you press ENTER at the Specify Second Point of Displacement prompt, the first point is treated as an *X,Y,Z* displacement.

# STYLE

## Quick Reference

Creates, modifies, or sets named text styles

Text toolbar: 

Format ➤ Text StyleAt the Command prompt, enter style.
**style** (or '**style** for transparent use)

The Text Style dialog box (page 1367) is displayed.

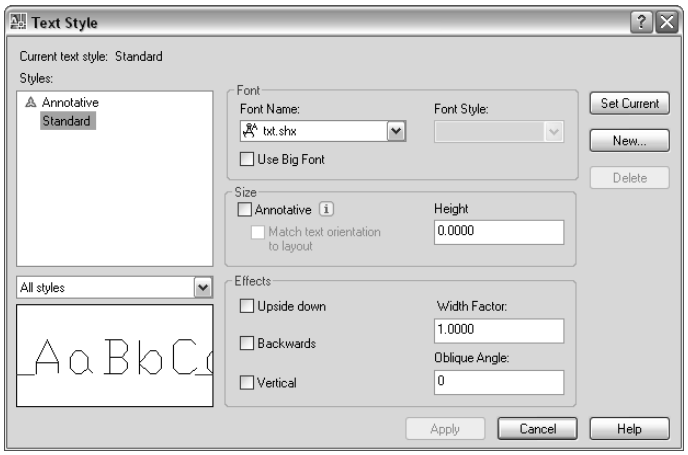
If you enter **-style** at the command prompt, options are displayed at the command prompt (page 1371).

# Text Style Dialog Box

## Quick Reference

Format ➤ Text StyleAt the Command prompt, enter style.
**style** (or '**style** for transparent use)

Creates, modifies, or sets named text styles.




### Current Text Style

Lists the current text style.

### Styles

Displays the list of styles in the drawing. The list contains defined style names and displays the current style that is selected by default. To change the current style, select another style from the list or choose New to create a new style. A

 icon before the style name indicates that the style is .

Style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign ($), underscore (_), and hyphen (-).

### Style List Filter

The drop-down list specifies whether all styles or only the styles in use are displayed in the styles list.

### Preview

Displays sample text that changes dynamically as you change fonts and modify the effects.

### Font

Changes the style's font.

**NOTE** If you change the orientation or font file of an existing text style, all text objects with that style use the new values when the drawing is regenerated.

**Font Name** Lists the font family name for all registered TrueType fonts and all compiled shape (SHX) fonts in the *Fonts* folder. When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font. See "Assign Text Fonts" in the *User's Guide*.

**Font Style** Specifies font character formatting, such as italic, bold, or regular. When Use Big Font is selected, this option changes to Big Font Name and is used to select a Big Font file name.

**Use Big Font** Specifies an Asian-language Big Font file. Use Big Font is available only if you specify an SHX file under Font Name. Only SHX files are valid file types for creating Big Fonts.

For more information on using Asian-language Big Fonts, see "Use Text Fonts for International Work" in the *User's Guide*.

### Size

Changes the size of the text.

**Annotative** Specifies that the text is . Click the information icon to learn more about the annotative objects.

**Match Text Orientation to Layout** Specifies that the orientation of the text in paper space viewports matches the orientation of the layout. This option is unavailable if the option is cleared.

**Height or Paper Text Height** Sets the text height based on the value you enter. If you enter **0.0**, the text defaults to a height of **0.2** each time you enter text using this style. Entering a height greater than **0.0** sets the text height for this style. TrueType fonts can be displayed at a smaller height than SHX fonts with the same height setting. If the annotative option is selected, sets the height that the text will display in paper space. See "Set Text Height" in the *User's Guide*.

### Effects

Modifies characteristics of the font, such as its height, width factor, and obliquing angle and whether it is displayed upside down, backwards, or vertically aligned.

**Upside Down** Displays the characters upside down.

**Backwards** Displays the characters backwards.

**Vertical** Displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for TrueType fonts.

**Width Factor** Sets the character spacing. Entering a value less than **1.0** condenses the text. Entering a value greater than **1.0** expands it.

**Oblique Angle** Sets the obliquing angle of the text. Entering a value between **-85** and **85** italicizes the text.

---

**NOTE** TrueType fonts using the effects described in this section might appear bold on the screen. Onscreen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.

---

### Set Current

Sets the style selected under Styles to current.

### New

Displays the New Text Style dialog box and automatically supplies the name " style*n*" (where *n* is the number of the supplied style) for the current settings. You can accept the default or enter a name and choose OK to apply the current style settings to the new style name.

### Delete

Deletes unused text styles.

### Apply

Applies style changes made in the dialog box to the current style and to the text of the current style in the drawing.

# -STYLE

## Quick Reference

If you enter **-style** at the command prompt, the following STYLE command prompts are displayed.

Enter name of text style (page 1371) or [? (page 1372)] <*current*>: *Enter a style name, enter* **?***, or press* ENTER

**Text Style Name** Specifies the text style name. Names can have up to 255 characters and can include letters, numbers, spaces, and any special character not used by Microsoft® Windows® or this program for other purposes. To define a style that uses Big Fonts, you can use long file names that do not contain commas. Commas are used to separate SHX files and Big Font files for defining a Big Font file. Pressing ENTER displays the following prompt (full font names are displayed only if the current font is a TrueType font):

Existing style = *current*:

Specify full font name or font file name (TTF or SHX) <*current*>:

Enter a TrueType font family name or an SHX font file name. If you do not enter a file name extension, this program searches for an SHX file. If the file is not located, Windows substitutes the first located registered TrueType font.

All long file names except those containing commas are accepted at the prompt. The comma is reserved for the Big Font naming convention: an SHX file followed by a comma (,), followed by the Big Font file name. A space is interpreted as part of the font name, not as a carriage return.

Entering a tilde (~) displays the Select Font File dialog box.

In the Select Font File dialog box, valid types include SHX and TTF. The character definitions of the selected font file are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font file.

Specify height of text or [Annotative] <*current*>: *Specify a distance, enter* **a***, or press* ENTER

If you enter **annotative**, you are prompted to create an text style.

Create annotative text style [Yes/No] <*current*>: *Enter* **y** *or* **n** *or press* ENTER

If you enter **yes** the current text style becomes annotative.

Match text orientation to layout? [Yes/No] <*current*>: *Enter* **y** *or* **n** *or press* ENTER

If you enter **yes** the current text style orientation in paper space viewports matches the layout.

Specify height of text or [Annotative] <*current*>: *Specify a distance or press* ENTER

If you enter a height of **0.0**, you are prompted for the text height each time you enter text using this style. Entering a height greater than **0.0** sets the Text Height (Non annotative), entering a height greater than **0.0** sets the Paper Text Height (Annotative), for this style.

Specify width factor <*current*>: *Specify a distance or press* ENTER

Entering a value less than **1.0** condenses the text. Entering a value greater than **1.0** expands it.

Specify obliquing angle <*current*>: *Specify an angle or press* ENTER

Entering a value between **-85** and **85** obliques the text.

Display text backwards? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

Display text upside-down? [Yes/No] <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

Vertical? <*current*>: *Enter* **y** *or* **n**, *or press* ENTER

Vertical is available only if the selected font supports dual orientation.

"*Current*" is now the current text style.

**?—List Text Styles** Lists the text styles available in the drawing.

Enter text style(s) to list <*>: *Enter the name of a style or press* ENTER

At the Enter Text Style(s) to List prompt, entering the name of a style displays the name, font file, height, width factor, obliquing angle, and generation of the style and exits the command. Entering an asterisk (*) or pressing ENTER displays the height, width factor, obliquing angle, and generation (whether text is drawn backwards, upside-down, vertically, or normally) of each style, and then exits the command.

# STYLESMANAGER

## Quick Reference

Displays the Plot Style Manager

File ➤ Plot Style ManagerAt the Command prompt, enter stylesmanager.
**stylesmanager**

The Plot Style Manager is displayed.

In the Plot Style Manager, you can double-click the Add-a-Plot-Style-Table wizard (page 1373) to add plot style tables. Double-click a plot style table (STB or CTB file) to start the Plot Style Table Editor (page 1373).

# Add-a-Plot-Style-Table Wizard

## Quick Reference

File ➤ Plot Style ManagerAt the Command prompt, enter stylesmanager.
**stylesmanager**

Adds new plot style tables. Plot style tables contain and define plot styles, which can be assigned to objects. When complete, the wizard produces an STB or CTB file depending on the type of table you are creating. You can edit these files in the Plot Style Table Editor (page 1373). To use your new plot style table, the selected options on the Plot and Publish tab of the Options dialog box (page 946) must be appropriate for the type of plot style table (named or color-dependent) you created.

You can create a plot style table from scratch, use an existing plot style table as a starting point, use settings from your AutoCAD Release 14 CFG file, or use settings from a PCP or PC2 file.

You can create a plot style table from scratch, use an existing plot style table as a starting point, or use settings from a PCP, PC2, or CFG file.

You can create either a named plot style table or a color-dependent plot style table. With a named plot style table, you can add and define plot styles as you like; the file name has the extension *.stb*. A color-dependent plot style table creates 255 plot styles based on color; the file name has the extension *.ctb*. You can specify whether you want to use a plot style table for new drawings or for pre-AutoCAD 2000 drawings when they are saved in a later format.

# Plot Style Table Editor

## Quick Reference

File ➤ Plot Style ManagerAt the Command prompt, enter stylesmanager.
**stylesmanager**

Modifies the plot styles in a plot style table. If the plot style table is attached to a layout or the Model tab, and you change a plot style, any objects that use that plot style are affected. If the plot style table is color-dependent, the file extension is CTB. If the plot style table is named, the file extension is STB. For information about plot style tables, see "Use Plot Styles to Control Plotted Objects" in the *User's Guide*.

Open the Plot Style Table Editor with any of the following methods:

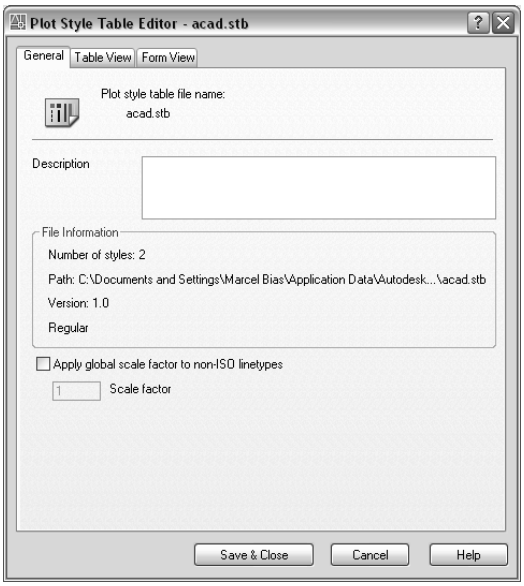
- Double-click a CTB or STB file in the Plot Style Manager.
- Right-click a CTB or STB file in the Plot Style Manager, and then choose Open from the shortcut menu.
- Choose Plot Style Table Editor from the Finish screen in the Add Plot Style Table wizard.
- In the Page Setup dialog box under Plot Style Table (Pen Assignments), select a plot style and click the Edit button.
- In the Current Plot Style and Select Plot Style dialog boxes, choose Editor.


- General (page 1374)
- Table View and Form View (page 1376)


## General Tab (Plot Style Table Editor)

### Quick Reference

Lists the plot style table file name, description, version number, location (path name), and table type. You can modify the description, and you can apply scaling to non-ISO linetypes and fill patterns.




**Plot Style Table File Name** Displays the name of the plot style table file you are editing.

**Description** Provides a description area for a plot style table.

**File Information** Displays information about the plot style table you are editing: number of plot styles, path, and version number of the Plot Style Table Editor.

**Apply Global Scale Factor to Non-ISO Linetypes** Scales all the non-ISO linetypes and fill patterns in the plot styles of objects controlled by this plot style table.

**Scale Factor** Specifies the amount to scale non-ISO linetypes and fill patterns.

**Delete R14 Color Mapping Table** Deletes color mapping tables that are used when you open pre-AutoCAD 2000 drawings. Named plot style tables that you create using *acadr14.cfg*, PCP, or PC2 files contain plot styles that are created from your AutoCAD Release 14 pen mappings. Color-dependent plot style tables also have color mapping tables. Color mapping tables are used to map plot styles to colors and thus to objects of each color when opening pre-AutoCAD 2000 drawings. While the color mapping table exists, you cannot add, delete, or rename plot styles in that plot style table.

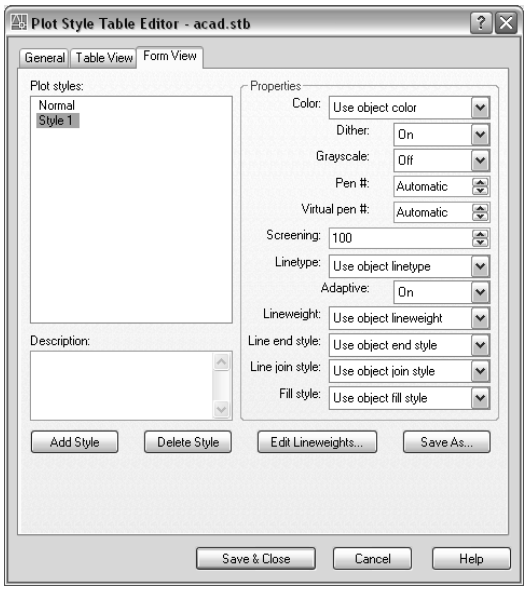
If you delete a color mapping table, plot styles cannot be automatically assigned to objects when pre-AutoCAD 2000 drawings are opened for the first time.

If you delete the mapping table, the plot style table becomes an ordinary plot style table and is no longer useful for applying plot styles to old drawings. It continues to be useful for new drawings.

## Table View and Form View Tabs (Plot Style Table Editor)

### Quick Reference

Lists all of the plot styles in the plot style table and their settings. Plot styles are displayed in columns from left to right. You can use either the Table View tab or the Form View tab to adjust plot style settings. In general, the Table View tab is convenient if you have a small number of plot styles. If you have a large number of plot styles, the Form view is more convenient because the plot style names are listed at the left and the properties of the selected style are displayed to the right. You don't have to scroll horizontally to view the style and its properties. The first plot style in a named plot style table is NORMAL and represents an object's default properties (no plot style applied). You cannot modify or delete the NORMAL style.



**Name** Displays the names of plot styles in named plot style tables. Plot styles in named plot style tables can be changed. Plot style names in color-dependent plot style tables are tied to object color and cannot be changed. The program accepts up to 255 characters for style names. You cannot have duplicate names within the same plot style table.

**Description** Provides a description for each plot style.

**Properties** Specifies the settings for the new plot style you are adding to the current plot style table.

**Color** Specifies the plotted color for an object. The default setting for plot style color is Use Object Color. If you assign a plot style color, the color overrides the object's color at plot time.

You can choose Select Color to display the Select Color dialog box (page 251) and select one of the 255 AutoCAD Color Index (ACI) colors, a true color, or a color from a color book. The color you specify is displayed in the plot style color list as Custom Color. If the plot device does not support the color you specify, it plots the nearest available color or, in the case of monochrome devices, black.

**Enable Dithering** Enables dithering. A plotter uses dithering to approximate colors with dot patterns, giving the impression of plotting more colors than available in the AutoCAD Color Index (ACI). If the plotter does not support dithering, the dithering setting is ignored.

Dithering is usually turned off in order to avoid false line typing that results from dithering of thin vectors. Turning off dithering also makes dim colors more visible. When you turn off dithering, the program maps colors to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available whether you use the object's color or assign a plot style color.

**Convert to Grayscale** Converts the object's colors to grayscale if the plotter supports grayscale. If you clear Convert to Grayscale, the RGB values are used for object colors. Dithering is available whether you use the object's color or assign a plot style color.

**Use Assigned Pen Number (Pen Plotters Only)** Specifies a pen to use when plotting objects that use this plot style. Available pens range from 1 to 32. If plot style color is set to Use Object Color, or you are editing a plot style in a color-dependent plot style table, you cannot change the assigned pen number; the value is set to Automatic.

If you specify 0, the field updates to read Automatic. The program determines the pen of the closest color to the object you are plotting using the information you provided under Physical Pen Characteristics in the Plotter Configuration Editor.

**Virtual Pen Number** Specifies a virtual pen number between 1 and 255. Many non-pen plotters can simulate pen plotters using virtual pens. For many devices, you can program the pen's width, fill pattern, end style, join style, and color/screening from the front panel on the plotter.

Enter 0 or Automatic to specify that the program should make the virtual pen assignment from the AutoCAD Color Index.

The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens. If this is the case, all the other style settings are ignored and only the virtual pen is used. If a non-pen plotter is not configured for virtual pens, then the virtual and physical pen information in the plot style is ignored and all the other settings are used.

You can configure your non-pen plotter for virtual pens under Vector Graphics on the Device and Document Settings tab in the PC3 Editor. Under Color Depth, select 255 Virtual Pens.

**Screening** Specifies a color intensity setting that determines the amount of ink placed on the paper while plotting. The valid range is 0 through 100. Selecting 0 reduces the color to white. Selecting 100 displays the color at its full intensity. In order for screening to work, the Enable Dithering option must be selected.

**Linetype** Displays a list with a sample and a description of each linetype. The default setting for plot style linetype is Use Object Linetype. If you assign a plot style linetype, the linetype overrides the object's linetype at plot time.

**Adaptive Adjustment** Adjusts the scale of the linetype to complete the linetype pattern. If you don't select Adaptive Adjustment, the line might end in the middle of a pattern. Turn off Adaptive Adjustment if linetype scale is important. Turn on Adaptive Adjustment if complete linetype patterns are more important than correct linetype scaling.

**Lineweight** Displays a sample of the lineweight as well as its numeric value. You can specify the numeric value of each lineweight in millimeters. The default setting for plot style lineweight is Use Object Lineweight. If you assign a plot style lineweight, the lineweight overrides the object's lineweight at plot time.

**Line End Style** Provides the following line end styles: Butt, Square, Round, and Diamond. The default setting for Line End Style is Use Object End Style. If you assign a line end style, the line end style overrides the object's line end style at plot time.

**Line Join Style** Provides the following line join styles: Miter, Bevel, Round, and Diamond. The default setting for Line Join Style is Use Object Join Style.

If you assign a line join style, the line join style overrides the object's line join style at plot time.

**Fill Style** Provides the following fill styles: Solid, Checkerboard, Crosshatch, Diamonds, Horizontal Bars, Slant Left, Slant Right, Square Dots, and Vertical Bar. The default setting for Fill Style is Use Object Fill Style. If you assign a fill style, the fill style overrides the object's fill style at plot time.

### Add Style

Adds a new plot style to a named plot style table.

The plot style is based on Normal, which uses an object's properties and doesn't apply any overrides by default. You must specify the overrides you want to apply after you create the new plot style. You cannot add a new plot style to a color-dependent plot style table; a color-dependent plot style table has 255 plot styles mapped to color. You also cannot add a plot style to a named plot style table that has a translation table.

### Delete Style

Deletes the selected style from the plot style table.

Objects assigned this plot style retain the plot style assignment but plot as Normal because the plot style is no longer defined in the plot style table. You cannot delete a plot style from a named plot style table that has a translation table, or from a color-dependent plot style table.

### Edit Lineweights

Displays the Edit Lineweights dialog box (page 1380). There are 28 lineweights available to apply to plot styles in plot style tables. If the lineweight you need doesn't exist in the list of lineweights stored in the plot style table, you can edit an existing lineweight. You can't add or delete lineweights from the list in the plot style table.

### Save As

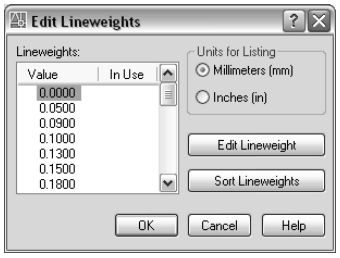
Displays the Save As dialog box and saves the plot style table to a new name.

## Edit Lineweights Dialog Box

### Quick Reference

File ➤ Plot Style ManagerAt the Command prompt, enter stylesmanager.
**stylesmanager**

Modifies the values of existing lineweights.




**Lineweights** Lists the lineweights in the plot style table. There are a total of 28 lineweights including Use Object Lineweight. You can modify existing lineweights, but you can't add or delete them. If you change a lineweight value, other plot styles that use the lineweight also change.

When you edit a lineweight value, it is rounded and displayed with a precision of four places past the decimal point. Lineweight values must be zero or a positive number. If you create a lineweight with a zero width, the line is plotted as thin as the plotter can create it. The maximum possible lineweight value is 100 millimeters (approximately four inches).

**Units for Listing** Specifies the units in which to display the list of lineweights. You can display the list of lineweights in inches or millimeters.

**Edit Lineweight** Makes the selected lineweight available for editing.

**Sort Lineweights** Sorts the list of lineweights by value. If you change lineweight values, choose Sort Lineweights to resort the list.

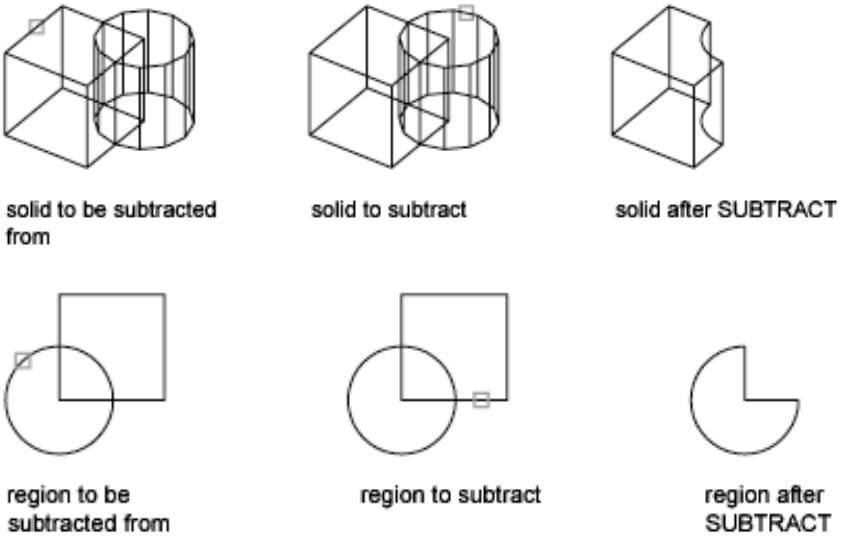
# SUBTRACT

### Quick Reference

Combines selected regions or solids by subtraction

Modeling 
Modify ➤ Solid Editing ➤ SubtractDoes not exist in the menus.
**subtract**
3D Make panel, Subtract
Select solids and regions to subtract from...
Select objects: *Use an object selection method and press* ENTER *when you finish*
Select solids and regions to subtract...
Select objects: *Use an object selection method and press* ENTER *when you finish*

Objects in the second selection set are subtracted from objects in the first selection set. A single new solid or region is created.




You can only subtract regions from other regions that are on the same plane. However, you can perform simultaneous SUBTRACT actions by selecting sets of regions on different planes. The program then produces separate subtracted regions on each plane. Regions for which there are no other selected coplanar regions are rejected.

# SUNPROPERTIES

### Quick Reference

Opens the Sun Properties window and sets the properties of the sun

View ➤ Render ➤ Light ➤ Sun PropertiesAt the Command prompt, enter sunproperties.
**sunproperties**
Expand the Light control panel and click Edit the Sun.

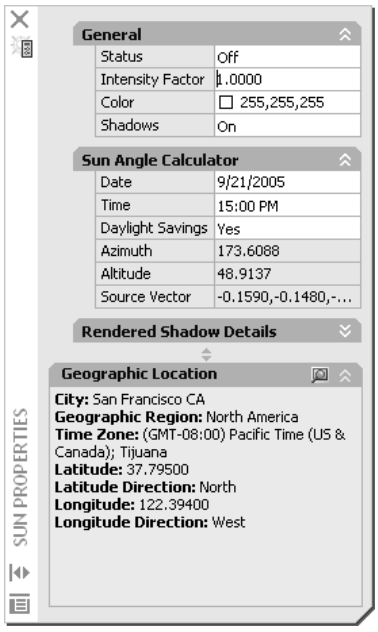
The Sun Properties window (page 1382)is displayed.

## Sun Properties Window

### Quick Reference

View ➤ Render ➤ Light ➤ Sun PropertiesDoes not exist in the menus.
**sunproperties**

Sets and modifies the properties of the sun.




### General

Sets the general properties of the sun.

**Status** Turns the sun on and off. If lighting is not enabled in the drawing, this setting has no effect.

**Intensity Factor** Sets the intensity or brightness of the sun. The range is from 0 (no light) to maximum. The higher the number, the brighter the light.

**Color** Controls the color of the light. Enter a color name or number or click Select Color to open the Select Color dialog box (page 251).

**Shadows** Turns display and calculation of shadows for the sun on and off. Turning shadows off increases performance.

### Sky Properties

The sky general properties are as follows:

**Status** Determines if the sky illumination is computed at render time. This has no impact on the viewport illumination or the background. It simply makes the sky available as a gathered light source for rendering. Note this does

not control the background. Values are Sky Off, Sky Background, Sky Background and Illumination. [Sky Off] is default.

**Intensity Factor** Provides a way to magnify the effect of the skylight. Values are 0.0-MAX. [1.0] is default.

**Haze** Determines the magnitude of scattering effects in the atmosphere. Values are 0.0-15.0. [0.0] is default.

### Horizon

This category of properties pertains to the appearance and location of the ground plane.

**Height** Determines the absolute position of the ground plane relative to world zero. This parameter represents a world-space length and should be formatted in the current length unit. Values are +-MAX. [0.0] is default.

**Blur** Determines the amount of blurring between ground plane and sky. Values are 0-10. [.1] is default.

**Ground Color** Determines the color of the ground plane by selecting a color from the drop-down list or select the Select Color dialog box (page 251) to make a color choice.

### Advanced

This category of properties pertains to various artistic effects.

**Night Color** Specifies the color of the night sky by selecting a color from the drop down list or select the Select Color dialog box (page 251) to make a color choice.

**Aerial Perspective** Specifies if aerial perspective is applied. Values are On/Off. [Off] is default.

**Visibility Distance** Specifies the distance at which 10% haze occlusion results. Values are 0.0-MAX. [10000.0] is default.

### Sun Disk Appearance

This category of properties pertains to the background only. They control the appearance of the sun disk.

**Disk Scale** Specifies the scale of the sun disk (1.0 = correct size).

**Glow Intensity** Specifies the intensity of the sun glow. Values are 0.0-25.0. [1.0] is default.

**Disk Intensity** Specifies the intensity of the sun disk. Values are 0.0-25.0. [1.0] is default.

### Sun Angle Calculator

Sets the angle of the sun.

**Date** Displays the current date setting.

**Time** Displays the current time setting.

**Daylight Saving** Displays the current setting for daylight saving time.

**Azimuth** Displays the azimuth, the angle of the sun along the horizon clockwise from due north. This setting is read-only.

**Altitude** Displays the altitude, the angle of the sun vertically from the horizon. The maximum is 90 degrees, or directly overhead. This setting is read-only.

**Source Vector** Displays the coordinates of the source vector, the direction of the sun. This setting is read-only.

### Rendered Shadow Details

Specifies the properties of the shadows.

**Type** Displays the setting for shadow type. This setting is read-only when display of shadows is turned off. The selections are Sharp, Soft (mapped) which display the Map size option and Soft (area) which displays the Samples option. Soft (area) is the only option for the sun in photometric workflow (LIGHTINGUNITS = 1 or 2).

**Map size (Standard lighting workflow only)** Displays the size of the shadow map. This setting is read-only when display of shadows is turned off. Values are 0-1000. [8] is default.

**Samples** Specifies the number of samples to take on the solar disk. This setting is read-only when display of shadows is turned off. Values are 0-1000. [8] is default.

**Softness** Displays the setting for the appearance of the edges of shadows. This setting is read-only when display of shadows is turned off. Values are 0-50.0. [1.0] is default.

**Geographic Location**

Displays the current geographic location settings. This information is read-only. When a city is not stored with latitude and longitude, the city does not appear in the list.

Use the Edit Geographic Location button to open the Geographic Location dialog box (page 604).

# SUNPROPERTIESCLOSE

## Quick Reference

Closes the Sun Properties window

**sunpropertiesclose**

The Sun Properties window (page 1382)is closed.

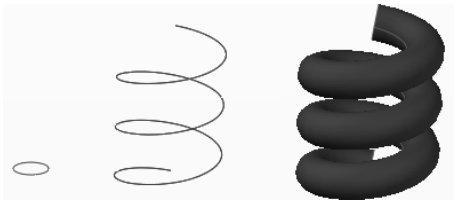
# SWEEP

## Quick Reference

Creates a 3D solid or surface by sweeping a 2D curve along a path

Modeling 
Draw ➤ Modeling ➤ SweepAt the Command prompt, enter sweep.
**sweep**
3D Make panel, Sweep

With the SWEEP command, you can create a new solid or surface by sweeping an open or closed planar curve (profile) along an open or closed 2D or 3D path. SWEEP draws a solid or surface in the shape of the specified profile along the specified path. You can sweep more than one object, but they all must lie on the same plane.



You can use the following objects and paths when creating a swept solid or surface:

| Objects that Can Be Swept | Objects that Can Be Used as a Sweep Path |
|---|---|
| Line | Line |
| Arc | Arc |
| Elliptical arc | Elliptical arc |
| 2D polyline | 2D polyline |
| 2D spline | 2D spline |
| Circle | Circle |
| Ellipse | Ellipse |
| Planar 3D face | 3D spline |
| 2D solid | 3D polyline |
| Trace | Helix |
| Region | Edges of solids or surface |
| Planar surface | |
| Planar faces of solid | |

**NOTE** You can select faces and edges on solids or surfaces by pressing and holding CTRL, and then selecting these subobjects.

The *DELOBJ* system variable controls whether the profile(s) and sweep path are automatically deleted when the solid or surface is created or whether you are prompted to delete the profile(s) and path.

You can select the objects to sweep before you start the command.

Current wire frame density: ISOLINES=4
Select objects to sweep: *Use an object selection method and press* ENTER *when you finish*
Select sweep path or [Alignment (page 1388)/Base Point (page 1388)/Scale (page 1388)/Twist (page 1389)]: *Select a 2D or 3D sweep path, or enter an option*

### Alignment

Specifies whether the profile is aligned to be normal to the tangent direction of the sweep path. By default, the profile is aligned.

Align sweep object perpendicular to path before sweep [Yes/No] <Yes>: *Enter* no *to specify that the profile is not be aligned or press* ENTER *to specify that the profile is aligned*

**NOTE** If the profile curve is not perpendicular (normal) to the tangent of the start point of the path curve, then the profile curve automatically aligns. Enter No at the alignment prompt to prevent this.

### Base Point

Specifies a base point for the objects to be swept. If the specified point does not lie on the plane of the selected objects, it is projected onto the plane.

Specify base point: *Specify a base point for the selection set*

### Scale

Specifies a scale factor for a sweep operation. The scale factor is uniformly applied to the objects that are swept from the start to the end of the sweep path.

Enter scale factor or [Reference] <1.0000>: *Specify a scale factor, enter* **r** *for the reference option, or press* ENTER *to specify the default value*

#### Reference

Scales the selected objects based on the length you reference by picking points or entering values.

Specify start reference length <1.0000>: *Specify a beginning length from which to scale the selected objects*
Specify end reference length <1.0000>: *Specify a final length to which to scale the selected objects*

#### Twist

Sets a twist angle for the objects being swept. The twist angle specifies the amount of rotation along the entire length of the sweep path.

Enter twist angle or allow banking for a non-planar sweep path [Bank] <n>: *Specify an angle value less than 360, enter* **b** *to turn on banking, or press* ENTER *to specify the default angle value*
Select sweep path [Alignment/Base point/Scale/Twist]: *Select a sweep path or enter an option*

Banking specifies whether or not the curve(s) being swept will bank naturally (rotate) along a 3D sweep path (3D polyline, 3D spline, or helix).

# SYSWINDOWS

## Quick Reference

Arranges windows and icons when the application window is shared with external applications

**syswindows**
Enter an option [Cascade (page 1389)/tile Horizontal (page 1389)/tile Vertical (page 1390)/Arrange icons (page 1390)]: *Enter an option*

#### Cascade

Overlaps windows, leaving title bars visible.

#### Tile Horizontal

Arranges windows in horizontal, nonoverlapping tiles.

### Tile Vertical

Arranges windows in vertical, nonoverlapping tiles.

### Arrange Icons

Arranges the window icons.

# T Commands

# 20

**In this chapter**

- TRACE
- TRANSPARENCY
- TRAYSETTINGS
- TREESTAT
- TRIM

# TABLE

## Quick Reference

Creates an empty table object in a drawing

Draw
Draw ➤ TableAt the Command prompt, enter table.
Tables panel, Table
**table**

The Insert Table dialog box (page 1393) is displayed.

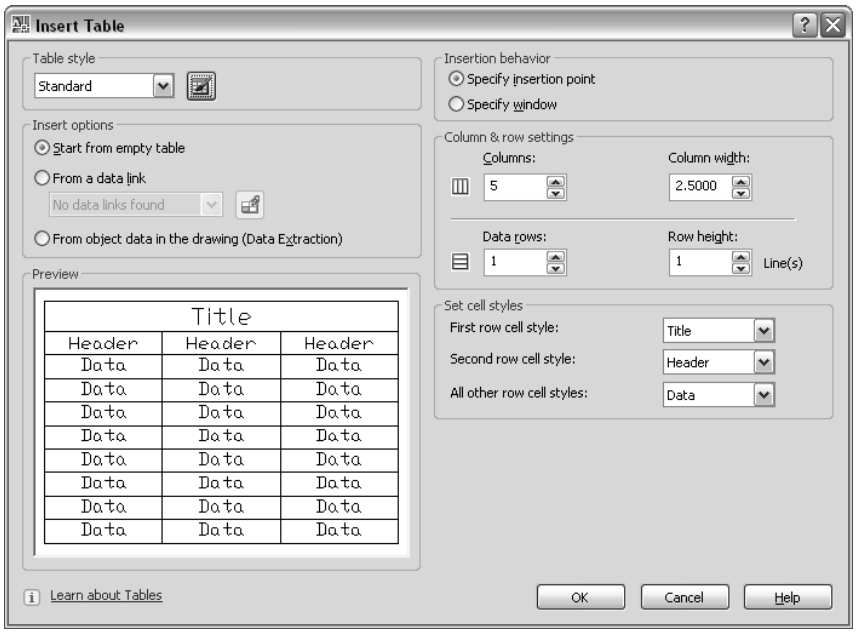
If you enter **-table** at the command prompt, options are displayed at the command prompt (page 1396).

# Insert Table Dialog Box

## Quick Reference

Draw
Draw ➤ TableAt the Command prompt, enter table.
Tables panel, Table
**table**

Inserts an empty table in the drawing.




### Table Style

Choose a table style from within the current drawing from which to create a table. You can create a new table style by clicking the button next to the drop-down list.

### Insert Options

Specifies the method for inserting your table.

**Start from Empty Table** Creates an empty table that can be filled in with data manually.

**Start from Data Link** Creates a table from data in an external spreadsheet.

**Start from Data Extraction** Launches the Data Extraction wizard.

### Preview

Displays an example of the current table style.

### Insertion Behavior

Specifies the location of the table.

**Specify Insertion Point** Specifies the location of the upper-left corner of the table. You can use the pointing device or enter coordinate values at the command prompt. If the table style sets the direction of the table to read from the bottom up, the insertion point is the lower-left corner of the table.

**Specify Window** Specifies a size and a location for the table. You can use the pointing device or enter coordinate values at the command prompt. When this option is selected, the number of columns and rows and the column width and row height depend on the size of the window and the column and row settings.

### Column & Row Settings

Set the number and size of columns and rows.

**Columns Icon** Indicates columns.

**Rows Icon** Indicates rows.

**Columns** Specifies the number of columns. When the Specify Window option is selected and you specify a column width, the Auto option is selected, and the number of columns is controlled by the width of the table. If a table style containing a starting table has been specified, then you can choose the number of additional columns you would like added to that starting table.

**Column Width** Specifies the width of the columns. When the Specify Window option is selected and you specify the number of columns, the Auto option is selected, and the column width is controlled by the width of the table. The minimum column width is one character.

**Data Rows** Specifies the number of rows. When the Specify Window option is selected and you specify a row height, the Auto option is selected, and the number of rows is controlled by the height of the table. A table style with a title row and a header row has a minimum of three rows. The minimum row height is one line. If a table style containing a starting table has been specified, then you can choose the number of additional data rows you would like added to that starting table.

**Row Height** Specifies the height of the rows in number of lines. The height of a line is based on the text height and the cell margin, which are both set in the table style. When the Specify Window option is selected and you specify the number of rows, the Auto option is selected, and the row height is controlled by the height of the table.

#### Set Cell Styles

For table styles that do not contain a starting table, specifies a cell style for rows in the new table.

**First Row Cell Style** Specifies a cell style for the first row in the table. The Title cell style is used by default.

**Second Row Cell Style** Specifies a cell style for the second row in the table. The Header cell style is used by default.

**All Other Row Cell Styles** Specifies a cell style for all other rows in the table. The Data cell style is used by default.

#### Table Options

For table styles that contain a starting table, specifies the table elements from the starting table that are retained upon insertion.

**Label Cell Text** Retains text from the Header or Title rows in the starting table in the newly-inserted table.

**Data Cell Text** Retains text from the Data rows in the starting table in the newly-inserted table.

**Blocks** Retains blocks from the starting table in the newly-inserted table.

**Retain Cell Style Overrides** Retains cell style overrides from the starting table in the newly-inserted table.

**Data Links** Retains data links from the starting table in the newly-inserted table.

**Fields** Retains fields from the starting table in the newly-inserted table.

**Formulas** Retains formulas from the starting table in the newly-inserted table.

## -TABLE

### Quick Reference

If you enter **-table** at the command prompt, the following TABLE command prompts are displayed.

Current table style: "Standard" Cell width: 2.5000 Cell height: 1 line(s)

Enter number of columns (page 1397) or [Auto (page 1398)/from Style (page 1398)/data Link (page 1399)] <5>:

### Number of Columns

Specifies the number of columns.

Enter number of rows (page 1397) or [Auto (page 1398)] <1>:

### Number of Rows

Specifies the number of rows.

Specify insertion point (page 1397) or [Style (page 1397)/Width (page 1397)/Height (page 1397)]:
**Insertion Point** Specifies the location of the upper-left corner of the table. You can use the pointing device or enter coordinate values at the command prompt. If the table style sets the direction of the table to read from the bottom up, the insertion point is the lower-left corner of the table.

**Width** Specifies a width for the table columns.

Enter cell width <2.50000>:

Specify insertion point or [Style/Width/Height]:

**Height** Specifies a height for the table rows.

Enter minimum cell height <1 line>:

Specify insertion point or [Style/Width/Height]:

**Style** Specifies a table style for the table. The specified table style must contain a starting table. To learn more about starting tables, see Work with Table Styles.

Enter table style name <*current*>

Specify insertion point or [Style/Width/Height]:

**Auto** Specifies a size and a location for the table. You can use the pointing device or enter coordinate values on the command line. When this option is selected, you can set the number of columns or the column width, but not both. The number of rows and the row height depend on the size of the window you specify.

Specify first corner or [Height]:

**First Corner** Specifies the location of the upper-left corner of the table. If the table style sets the direction of the table to read from the bottom up, the insertion point is the lower-left corner of the table.

Specify second corner:

**Height** Specifies a height for the table rows.

Enter minimum cell height <1 line>:

Specify first corner or [Height]:

### Auto

Specifies a size and a location for the table. You can use the pointing device or enter coordinate values at the command prompt. When this option is selected, you can set the number of columns or the column width, but not both. The number of rows and the row height depend on the size of the window you specify.

Enter number of rows (page 1398) or [Auto (page 1398)] <1>:

**Number of Rows** Specifies the number of rows.

Specify first corner or [Width]:

Specify second corner:

**Auto** Specifies a size and a location for the table.

Specify first corner or [Width]:

Specify second corner:

### From Style

Specifies a table style for creating the table. You can use the pointing device or enter coordinate values at the command prompt. The specified table style must contain a starting table. For more information about starting tables, see Work with Table Styles.

Specify insertion point (page 1398) or [Style (page 1398)/Rows (page 1399)/Columns (page 1399)/Options (page 1399)]:

**Insertion Point** Specifies the location of the upper-left corner of the table. You can use the pointing device or enter coordinate values at the command prompt. If the table style sets the direction of the table to read from the bottom up, the insertion point is the lower-left corner of the table.

Specify insertion point or [Style/Rows/Columns/Options]:

**Style** Specifies a table style for the table.

Enter the name of a table style or [?] <Standard>:

You are prompted if the table style you chose does not contain a starting table.

Table style does not contain a table.

Entering [?] displays a list of table styles available in your drawing.

**Rows** Specifies the number of rows you want to add to the table stored in the specified table style. The rows are added to the rows already in the specified table.

Enter number of additional rows <0>:

**Columns** Specifies the number of columns you want to add to the table stored in the specified table style. The columns are added to the columns already in the specified table.

Enter number of additional columns <0>:

### Options

Specifies special formatting options that can be inserted in the table.

Enter an insert option [Label text (page 1399)/Data text (page 1399)/Formulas (page 1399)/fIelds (page 1399)/data liNks (page 1399)/Blocks (page 1399)/cell style Overrides (page 1399)] <exit>:

**Label Text** Retains rows with a cell type of Label found in the table style's starting table. The cell type is set in the Properties palette. The Header and Title cell styles use the Label cell type by default.

**Data Text** Retains rows with a cell type of Data found in the table style's starting table. The cell type is set in the Properties palette. The Data cell style uses the Data cell type property by default.

**Formulas** Retains formulas found in the specified table style's starting table.

**Fields** Retains fields found in the specified table style's starting table. For more information about fields, see Use Fields in Text.

**Data Links** Retains data links found in the specified table style's starting table. For more information about data links, see Link a Table to External Data.

**Blocks** Retains blocks found in the specified table style's starting table.

**Cell Style Overrides** Retains cell style overrides found in the specified table style's starting table. For more information about cell style overrides, see Work with Table Styles.

### Data Link

Specifies a data link from which a table is created. You are prompted if the drawing does not contain any data links.

No data links exist in the current drawing.
To create a data link use the DATALINK (page 326) command.

Enter the name of a data link found in the current drawing.

Enter the name of a data link or [?]:

Entering [?] displays a list of data links available in your drawing.

You are prompted if the drawing does not contain the specified data link.

Data link not found.

After you specify a valid data link, specify an insertion point for the table that will be created from the link. You can use the pointing device or enter coordinate values at the command prompt.

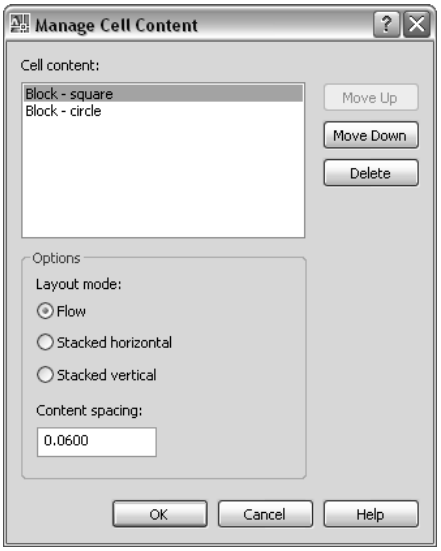
Specify insertion point:

# Manage Cell Content Dialog Box

## Quick Reference

Right-click while editing any single cell, and click Manage Cell Content.
Table

Displays the content of the selected cell. You can change the order of cell content as well as change the direction in which cell content will appear.



#### Cell Content List Box

Lists all text and/or blocks in the selected cell in order of appearance. Text is indicated with the label Table Cell Text. Blocks are indicated with Block preceding the name of the block.

#### Content Order Buttons

With the buttons to the right of the Current Content List Box, you can change the position of the highlighted content up or down, or remove content from the cell completely.

**Move Up** Moves the selected list box content up in the display order.

**Move Down** Moves the selected list box content down in the display order.

**Delete** Removes the selected list box content from the table cell.

#### Layout Mode Options

Changes the direction in which cell content will appear.

**Flow** Places cell content based on the width of the cell.

**Stacked Horizontal** Places cell content horizontally, regardless of cell width.

**Stacked Vertical** Places cell content vertically, regardless of cell height.

**Content Spacing** Determines the spacing between text and/or blocks within the cell.

# TABLEDIT

## Quick Reference

Edits text in a table cell

Double-click inside a table cell.
With a table cell selected, right-click and click Edit Cell Text.
**tabledit**
Pick a table cell: *Click inside a table cell, and enter text or use the Text Formatting toolbar or the Options shortcut menu to make changes*

# TABLEEXPORT

### Quick Reference

Exports data from a table object in CSV file format

With a table selected, right-click and click Export.
**tableexport**

A standard file selection dialog box (page 931) is displayed. Table data is exported in the comma-separated (CSV) file format. All formatting of the table and its text is lost.

# TABLESTYLE

### Quick Reference

Defines a new table style

Styles 
Format ➤ Table StyleAt the Command prompt, enter tablestyle.
Tables panel, Table Style
**tablestyle**

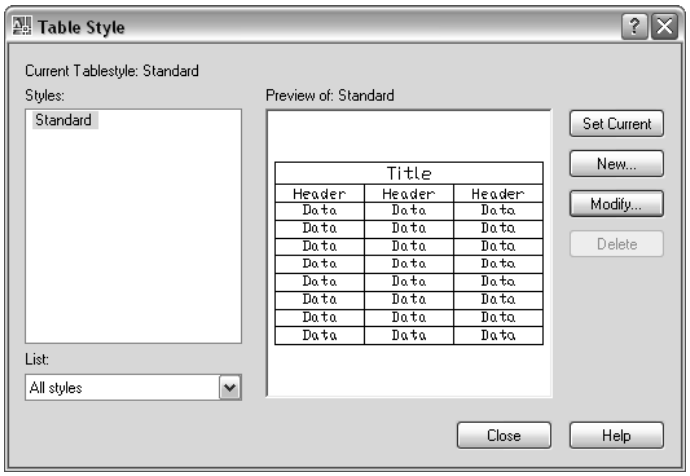
The Table Style dialog box (page 1402) is displayed.

## Table Style Dialog Box

### Quick Reference

Styles
Format ➤ Table StyleAt the Command prompt, enter tablestyle.
**tablestyle**

Sets the current table style and creates, modifies, and deletes table styles.




### Current Table Style

Displays the name of the table style that is applied to tables you create. The default table style is STANDARD.

### Styles

Displays a list of table styles. The current style is highlighted.

### List

Controls the contents of the Styles list.

**All Styles** Displays all table styles.

**Styles in Use** Displays only the table styles that are referenced by tables in the current drawing.

### Preview Of

Displays a preview image of the style that is selected in the Styles list.

### Set Current

Sets the table style selected in the Styles list as the current style. All new tables are created using this table style.

### New

Displays the Create New Table Style dialog box (page 1404), in which you can define new table styles.

### Modify

Displays the Modify Table Style dialog box (page 1405), in which you can modify table styles.

### Delete

Deletes the table style selected in the Styles list. A style that is being used in the drawing cannot be deleted.

## Create New Table Style Dialog Box

### Quick Reference

Styles
Format ➤ Table StyleAt the Command prompt, enter tablestyle.
**tablestyle**

Specifies a name for the new table style and specifies the existing table style on which the new table style will be based.


**New Style Name** Names the new table style.

**Start With** Specifies an existing table style whose settings are the default for the new table style.

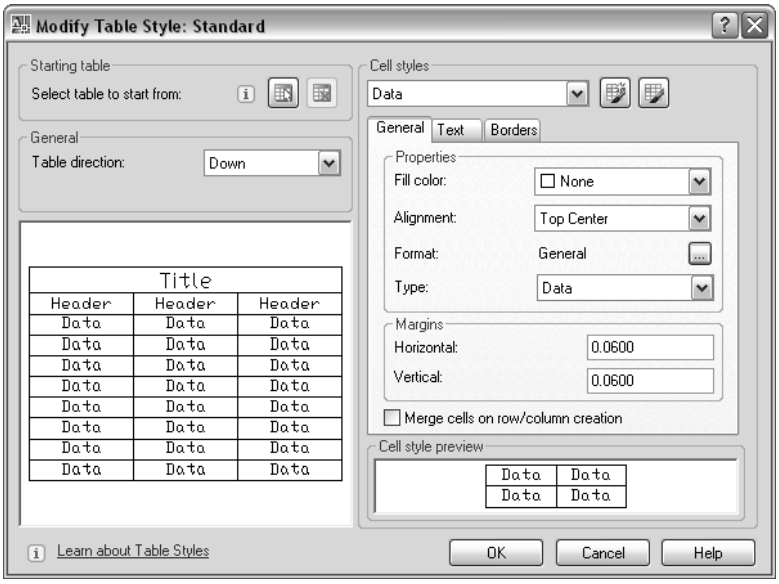
**Continue** Displays the New Table Style dialog box (page 1405), in which you define the new table style.

# New and Modify Table Style Dialog Boxes

## Quick Reference

Styles
Format ➤ Table StyleAt the Command prompt, enter tablestyle.
**tablestyle**

Defines a new table style or modifies an existing table style. The options in each dialog box are the same.




### Starting Table

Allows you to specify a table in your drawing to use as an example for formatting this table style. Once you select a table, you can specify the structure and contents you want copied from that table to the table style.

With the Remove Table icon, you can remove a table from the current specified table style.

### General

Changes the direction of the table.

**Table Direction** Sets the direction of a table. Down creates a table that reads from top to bottom. Up creates a table that reads from bottom to top.

- *Down*: The title row and the column heads row are at the top of the table. When you click Insert Rows and click Below, the new row is inserted below the current row.
- *Up*: The title row and the column heads row are at the bottom of the table. When you click Insert Rows and click Below, the new row is inserted above the current row.

### Preview

Displays an example of the effect of the current table style settings.

### Cell Styles

Defines a new cell style or modifies an existing cell style. You can create any number of cell styles.

**Cell Style Menu** Displays cell styles already found within the table.

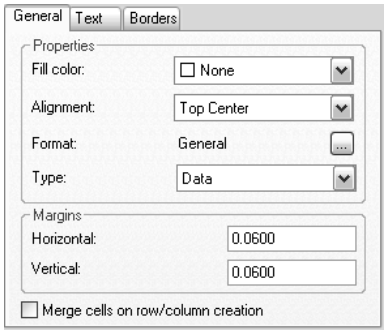
**Create Cell Styles Button** Launches the Create New Cell Style dialog box. (page 1409)

**Manage Cell Styles Button** Launches the Manage Cell Styles dialog box (page 1410).

### Cell Style Tabs

Set the appearance of the data cells, the cell text, and the cell borders, depending on which tab is active: General tab, Text tab, or Borders tab.

#### General Tab



### Properties

**Fill Color** Specifies the background color of the cell. The default is None.

You can choose Select Color to display the Select Color dialog box (page 251).

**Alignment** Sets justification and alignment for the text in the table cell. Text is middle-, top-, or bottom-aligned with respect to the top and bottom borders of the cell. Text is center-justified, left-justified, or right-justified with respect to the left and right borders of the cell.

See "Justify Multiline Text" in the *User's Guide* for an illustration of the nine options.

**Format** Sets data type and formatting for the Data, Column Heading, or Title rows in a table. Clicking this button displays the Table Cell Format dialog box (page 1411), where you can further define formatting options.

**Type** Specifies the cell style as either a label or data.

### Margins

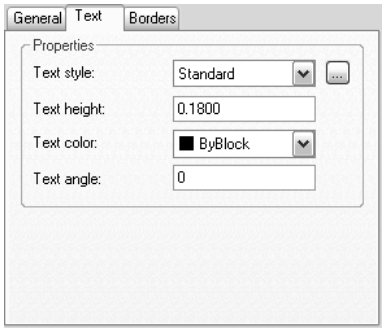
Controls the spacing between the border of the cell and the cell content. The cell margin settings apply to all cells in the table. The default setting is 0.06 (imperial) and 1.5 (metric).

**Horizontal** Sets the distance between the text or block in the cell and the left and right cell borders.

**Vertical** Sets the distance between the text or block in the cell and the top and bottom cell borders.

**Merge cells on row/column creation** Merges any new row or column created with the current cell style into one cell. You can use this option to create a title row at the top of your table.

#### Text Tab



**Text Style** Lists all the text styles in the drawing.

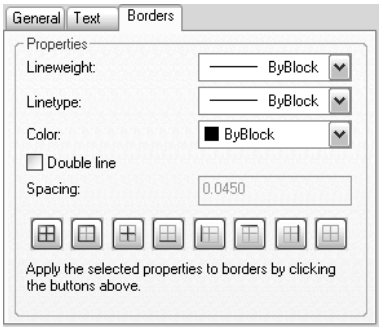
The [...] button displays the Text Style dialog box (page 1367), where you can create a new text style.

**Text Height** Sets the text height. The default text height for data and column head cells is 0.1800. The default text height for the table title is 0.25.

**Text Color** Specifies the color of the text. Choose Select Color at the bottom of the list to display the Select Color dialog box (page 251).

**Text Angle** Sets the text angle. The default text angle is 0 degrees. You can enter any angle between -359 and +359 degrees.

**Borders Tab**




**Lineweight** Sets the lineweight to be applied to the borders you specify by clicking a border button. If you use a heavy lineweight, you may have to increase the cell margins.

**Linetype** Sets the linetype to be applied to the borders you specify by clicking a border button. Standard linetypes of ByBlock, ByLayer, and Continuous are displayed, or you can choose Other to load a custom linetype.

**Color** Sets the color to be applied to the borders you specify by clicking a border button. Choose Select Color to display the Select Color dialog box (page 251).

**Double Line** Displays table borders as double lines.

**Spacing** Determines the spacing for double line borders. The default spacing is 0.1800.

**Border Buttons**

Controls the appearance of the borders of the cells. The border properties are lineweight and color of the gridlines.

**All Borders** Applies the border properties settings to all borders for the specified cell style.

**Outside Border** Applies the border properties settings to the outside border for the specified cell style.

**Inside Border** Applies the border properties settings to the inside border for the specified cell style.

**Bottom Border** Applies the border properties settings to the bottom borders for the specified cell style.

**Left Border** Applies the border properties settings to the left borders for the specified cell style.

**Top Border** Applies the border properties settings to the top borders for the specified cell style.

**Right Border** Applies the border properties settings to the right borders for the specified cell style.

**No Border** Hides borders for the specified cell style.

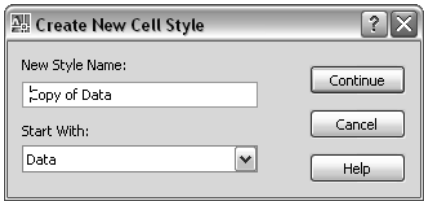
#### Cell Style Preview

Displays an example of the effect of the current table style settings.

# Create New Cell Style Dialog Box

## Quick Reference

Styles




Format ➤ Table StyleAt the Command prompt, enter tablestyle.
**tablestyle**

Specifies a name for the new cell style and specifies the existing cell style on which the new cell style will be based.

**New Style Name** Names the new cell style.

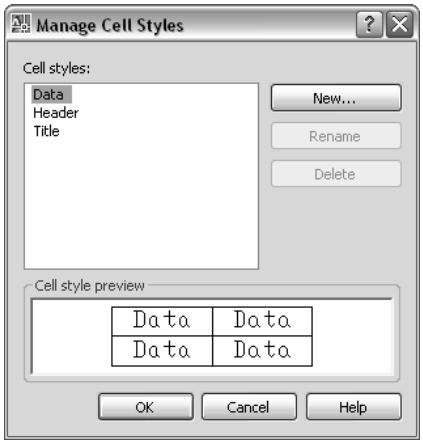
**Start With** Specifies an existing cell style whose settings are the default for the new cell style.

**Continue** Returns you to the New Table Style dialog box (page 1405), in which you define the new cell style.

# Manage Cell Styles Dialog Box

## Quick Reference

Format ➤ Table Style




### tablestyle

Displays all cell styles within the current table style and allows you to create or delete a cell style.

### Cell Styles

Lists all cell styles contained within the current table style. The cell styles Title, Header, and Data are always contained within any table style, and cannot be deleted or renamed.

**New** Displays the Create New Cell Style dialog box (page 1409). From here, you can create a new cell style to be contained within the current table style.

**Rename** Allows you to give a new name to the selected cell style. The Title, Header, and Data cell styles cannot be renamed.

**Delete** Allows you to delete the selected cell style.

### Cell Style Preview

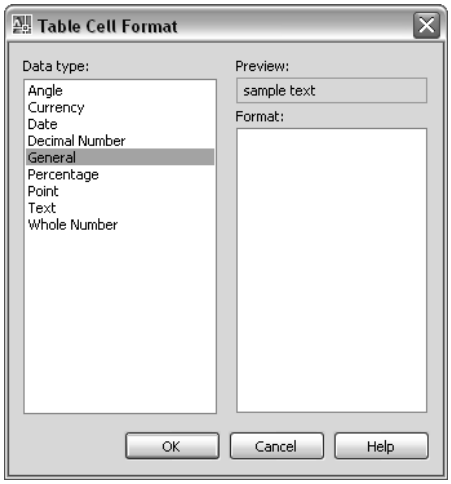
Displays an example of the effect of the current cell style settings.

# Table Cell Format Dialog Box

## Quick Reference

Format ➤ Table StyleAt the Command prompt, enter tablestyle.
**tablestyle**

Defines the formatting for cells in your table. The options displayed here change based on the selected data type and format you select.




### Data Type

Displays a list of data types (Angle, Date, Decimal Number, and so on) that you can format for table rows.

**Preview** Displays a preview of the option you selected in the Format list.

**Format** Depending on the data type you select, displays a list of relevant format types. For example, if you selected Angle as the data type, options such as Decimal Degrees, Grads, Radians, and so on are displayed.

**Precision** For Angle, Decimal Number, and Points data types only, sets the precision for applicable formats. For example, if you select Angle as the data type and Radians as the format type, options such as Current Precision, 0.0r, 0.00r, 0.000r, and so on are displayed.

**List Separator** For a Point data type only, displays a list of options (comma, semicolon, or colon) that you can use to separate list items.

**Symbol** For Currency data types only, displays a list of currency symbols that you can use.

**Append Symbol** In Currency data types, places the currency symbol after the number. In the Percentage data types, places the percent symbol after the number.

**Negative Numbers** For Currency data types only, lists options for displaying negative numbers.

**X, Y, and Z Coordinates** For a Point data type only, filters X, Y, or Z coordinates.

**Additional Format** For Angle, Decimal Number, Point, and Whole Number data types only, opens the Additional Format dialog box (page 1412), where you set additional formatting options for table cells.

**Examples** For the Date data type only, displays a list of date display options for the date option you selected in the Format field. Click a date in the Format field to see an example.

# Additional Format Dialog Box

## Quick Reference

Format ➤ Table Style
Right-click while any text command is active, and click Insert Field.
Format ➤ Table StyleAt the Command prompt, enter tablestyle.

Provides additional formatting options for fields and table cells.

### Current Value

Displays the value in base drawing units.

### Preview

Displays updates to the format as you change the conversion factor and other settings.

### Conversion Factor

Specifies the conversion factor to use on the the current value. The default is 1 for no conversion.

### Additional Text

Specifies a prefix or a suffix for the value.

### Number Separators

Specifies a decimal separator and the formatting for numbers over 1000.

**Decimal** Specifies the separator for decimal values. Select a period, a comma, or a space.

**Thousands** Inserts a comma to group thousands in a field value.

### Zero Suppression

Controls the suppression of leading and trailing zeros, and of feet and inches that have a value of zero.

**Leading** Suppresses leading zeros in all decimal values. For example, 0.5000 becomes .5000.

**Trailing** Suppresses trailing zeros in all decimal values. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

**0 Feet** Suppresses the feet portion of a feet-and-inches value when the distance is less than one foot. For example, 0'-6 1/2" becomes 6 1/2".

**0 Inches** Suppresses the inches portion of a feet-and-inches value when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

# TABLET

## Quick Reference

Calibrates, configures, and turns on and off an attached digitizing tablet

Tools ➤ TabletDoes not exist in the menus.
**tablet**
Enter an option [ON (page 1414)/OFF (page 1414)/CAL (page 1414)/CFG (page 1417)]:

### On

Turns on Tablet mode. Setting the *TABMODE* system variable to 1 also turns on Tablet mode. Pressing CTRL+T on some systems turns Tablet mode on and off.

### Off

Turns off Tablet mode. Setting the *TABMODE* system variable to 0 also turns off Tablet mode. Pressing CTRL+T on some systems turns Tablet mode on and off.

### Cal

Calibrates the digitizer with a paper drawing or photograph, creating the tablet transformation, which is the mapping of points from the tablet to the coordinate drawing system. Calibration is digitizing points on the drawing and mapping them to their real coordinates.

Calibration can be performed in model space or paper space. The Cal option turns on Tablet mode in the space in which the tablet is calibrated. When the space is changed, the Cal option turns off Tablet mode.

The paper should be flat (with no bumps or wrinkles) and securely fastened to the digitizing tablet. The paper can be oriented at any angle.

Digitize point #1: *Digitize a point on the paper drawing*
Enter coordinates for point #1: *Specify an X,Y coordinate at the digitized point*
Digitize point #2: *Digitize a point on the paper drawing*
Enter coordinates for point #2: *Specify an X,Y coordinate at the digitized point*
Digitize point #3 (or ENTER to end): *Digitize a point on the tablet or press* ENTER
Enter coordinates for point #3: *Specify an X,Y coordinate in the drawing*

The points you enter cannot be duplicates. The points need not be the origin on either axis, and you can enter as many points as you like. The more points you enter, the more accurate the drawing will be.

If you enter only two points, the program automatically computes an orthogonal transformation. If it is successful, the command ends.

If you enter three or more points, the program computes the transformation in each of the three transformation types (Orthogonal, Affine, and Projective) to determine which best fits the calibration points. If you enter more than four points, computing the best-fitting projective transformation can take a long time. You can cancel the process by pressing ESC.

When the computations are complete, the program displays a table with the number of calibration points and a column for each transformation type.

If there have been no failures of projection transformation, the program prompts you to choose a transformation type.

Enter transformation type [Orthogonal/Affine/Projective/Repeat table] <Repeat>: *Enter an option or press* ENTER

Only transformation types for which the outcome was Success, Exact, or Canceled are included in this prompt. A projective transformation can be specified even if it was canceled. The program uses the result computed at the time you canceled.

**Orthogonal** Specifies translation, uniform scaling, and rotation with two calibration points.

Use Orthogonal for dimensionally accurate paper drawings and paper drawings in which the portion to be digitized is long and narrow, with most points confined to single lines.

---

**NOTE** You must specify the lower-left point location before specifying the upper-right point location.

---

**Affine** Specifies arbitrary linear transformation in two dimensions consisting of translation, independent *X*- and *Y*-scaling, rotation, and skewing with three calibration points.

Use Affine when horizontal dimensions in a paper drawing are stretched with respect to vertical dimensions, and lines that are supposed to be parallel actually are parallel.

The RMS (root mean square) error reported after calibration measures how close the program has come to making a perfect fit. Affine should be used if the RMS is small.

**Projective** Specifies a transformation equivalent to a perspective projection of one plane in space onto another plane with four calibration points. A projective transformation provides a limited form of what cartographers call *rubber sheeting*, in which different portions of the tablet surface are stretched by varying amounts. Straight lines map into straight lines. Parallel lines do not necessarily stay parallel.

Projective transformation corrects parallel lines that appear to converge.

**Repeat Table** Redisplays the computed table, which rates the transformation types.

### Transformation Table

Reports the number of calibration points and provides information about each transformation type.

### Outcome of Fit

Reports the outcome of fit for each of the transformation types. If the outcome of fit is not Success or Exact for any of the transformation types, the program reports failure of the entire calibration process and ends the command. The remaining entries in each column are blank unless Outcome of Fit is Success.

**Exact** Indicates the correct number of points for a valid transformation.

**Success** Indicates more than enough points. The program succeeded in fitting a transformation to the data.

**Impossible** Indicates not enough points.

**Failure** Indicates enough points, but the program was unable to fit a transformation to the points, usually because some points were colinear or coincident.

**Canceled** Indicates that the fit process was canceled. This outcome occurs only with the projective transformation.

### RMS Error

Reports the RMS (root mean square) error, which measures how close the program has come to finding a perfect fit. The goal is the smallest RMS error.

### Standard Deviation

Reports the standard deviation of the residuals. If it is near zero, the residual at each calibration point is about the same.

### Largest Residual/At Point

Reports the point at which the mapping is least accurate. The residual is the distance between where the point was mapped during transformation and where it would be mapped if the fit were perfect. The distance is given in the current linear units.

### Second-Largest Residual/At Point

Reports the point at which the mapping is second-least accurate. The residual is the distance between where the point was mapped during transformation and where it would be mapped if the fit were perfect. The distance is given in the current linear units.

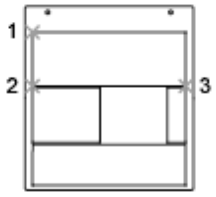
## Cfg

Designates or realigns the tablet menu areas or designates a small portion of a large tablet as a screen pointing area.

Enter number of tablet menus desired (0-4) <*current*>: *Enter a value or press* ENTER

If tablet menus are in use and the same number of menus is selected, the following prompts are displayed:

Do you want to realign tablet menus? [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER
Digitize upper-left corner of menu area *n*: *Digitize a point (1)*
Digitize lower-left corner of menu area *n*: *Digitize a point (2)*
Digitize lower-right corner of menu area *n*: *Digitize a point (3)*




In the prompt, *n* is the menu number (1-4).

The printed menu form must be affixed to the tablet surface, and the requested points must be digitized. The set of three points must form a 90-degree angle. Tablet menu areas can be skewed at any angle.

Enter the number of columns for menu area *n*: *Enter a positive value*
Enter the number of rows for menu area *n*: *Enter a positive value*

After all interaction concerning tablet menus is complete, the following prompt is displayed:

Do you want to respecify the screen pointing area? [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER

If you enter **y**, the following prompts are displayed:

Digitize lower-left corner of the pointing area: *Digitize a point*
Digitize upper-right corner of the pointing area: *Digitize a point*

A small portion of the tablet's surface is designated as the fixed screen pointing area. The following prompt is displayed:

Do you want to specify the Floating Screen Pointing area? [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER

If you responded **y** to the previous prompt, respond to the following prompt:

Do you want the Floating Screen Area to be the same size as the Fixed Screen Pointing Area? [Yes/No] <Y>: *Enter* **y** *or* **n** *or press* ENTER

If you responded **n** to the previous prompt, respond to the following prompts:

Digitize lower-left corner of the Floating Screen pointing area: *Digitize a point*
Digitize upper-right corner of the Floating Screen pointing area: *Digitize a point*

Respond to the following prompt:

Would you also like to specify a button to toggle the Floating Screen Area? [Yes/No] <N>: *Enter* **y** *or* **n** *or press* ENTER

If you responded **y** to the previous prompt, respond to the following prompt:

Press any non-pick button on the digitizer puck that you wish to designate as the toggle for the Floating Screen Area

# TABSURF

## Quick Reference

Creates a tabulated mesh from a path curve and a direction vector

Draw ➤ Modeling ➤ Meshes ➤ Tabulated MeshAt the Command prompt, enter tabsurf.
**tabsurf**
Select object for path curve:

The path curve defines the approximated surface of the polygon mesh. It can be a line, arc, circle, ellipse, or 2D or 3D polyline. The mesh is drawn starting at the point on the path curve closest to the selection point.

Select object for direction vector: *Select a line or open polyline*


Only the first and last points on a polyline are considered, and intermediate vertices are ignored. The direction vector indicates the direction and length of the shape to be extruded. The end selected on the polyline or line determines the direction of the extrusion. The original path curve is drawn with wide lines to help you visualize how the direction vector dictates the construction of a tabulated mesh.


TABSURF constructs a 2 by *n* polygon mesh, where *n* is determined by the *SURFTAB1* system variable. The *M* direction of the mesh is always 2 and lies along the direction vector. The *N* direction lies along the path curve. If the path curve is a line, arc, circle, ellipse, or spline-fit polyline, tabulation lines are drawn that divide the path curve into intervals of equal size set by SURFTAB1. If the path curve is a polyline that has *not* been spline fit, tabulation lines are drawn at the ends of straight segments, and each arc segment is divided into intervals set by SURFTAB1.


# TARGETPOINT

## Quick Reference

Creates a target point light

**targetpoint**
Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*
Specify target location <0,0,-10>: *Enter coordinate values or use the pointing device*

If the LIGHTINGUNITS system variable is set to 0, the following prompt is displayed:

Enter an option to change [Name (page 1420)/Intensity (page 1420)/Status (page 1420)/shadoW (page 1421)/Attenuation (page 1422)/Color (page 1423)/eXit (page 1423)] <eXit>:

If the LIGHTINGUNITS system variable is set to 1 or 2, the following prompt is displayed:

Enter an option to change [Name (page 1420)/Intensity factor (page 1420)/Status (page 1420)/Photometry (page 1420)/shadoW (page 1421)/Attenuation (page 1422)/filterColor (page 1423)/eXit (page 1423)] <eXit>:

**NOTE** When the LIGHTINGUNITS system variable is set to 1 or 2, the Attenuation option has no affect on the creation of the light. It is only maintained for scripting compatibility.

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity/Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00-max float) <1.0000>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf] <On>:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived power per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i**, you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*> :

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFtmapped/softsAmpled] <Sharp>:
**Off** Turns off the display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory that should be used to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness to use to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Linear, Disk, Rect, Sphere, Cylinder] <Sphere>:

Specify the sample size by entering **a** .

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v.**

Enter Shape Visibility [Yes/No]<No>:

### Attenuation

Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit]<eXit>:
**Attenuation Type** Controls how light diminishes over distance. The farther away an object is from a point light, the darker the object appears. Attenuation is also known as decay.

Enter attenuation type [None/Inverse linear/inverse Squared] <Inverse linear>:

- None. Sets no attenuation. Objects far from the point light are as bright as objects close to the light.
- Inverse Linear. Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the point light; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.

- Inverse Squared. Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the point light; at a distance of 4 units, light is one sixteenth as strong.

**Use Limits** Specifies whether to use limits or not.

Limits [oN/oFf] <Off>:

**Attenuation Start Limit** Specifies the point where light starts as an offset from the center of the light. The default is 0.

Specify start limit offset <1.0000>:

---

**NOTE** Attenuation start limits and end limits are not supported by the OpenGL driver (*wopengl9.hdi*), but the Direct 3D driver (*direct3d9.hdi*) does support end limits for attenuation but does not support start limits. To identify your driver, enter **3dconfig**, and click Manual Tune. Look at the selected Driver Name in the Manual Performance Tuning dialog box.

---

**Attenuation End Limit** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point. Setting an end limit increases performance where the effect of lighting is so minimal that the calculations are wasted processing time.

Specify end limit offset <10.0000>:

### Color/Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]<255,255,255>:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L) <0,0,100>:

**Color Book** Specifies a color from a color book.

Enter Color Book name:

### Exit

Exits the command.

# TASKBAR

## Quick Reference

Controls how drawings are displayed on the Windows taskbar

**taskbar**
Enter new value for Taskbar <0>: *Enter* **1** *to display multiple open drawings as separate items on the Windows taskbar*

The default setting, 0, displays only the name of the current drawing on the Windows taskbar.

# TEXT

## Quick Reference

Creates a single-line text object

Draw ➤ Text ➤ Single Line TextAt the Command prompt, enter dtext.
**text**
Current text style: <*current*> Current text height: <*current*> Annotative: <*current*> Specify start point (page 1425) of text or [Justify (page 1425)/Style (page 1430)]: *Specify a point or enter an option*

The TEXT command create a single-line text object. It displays a simplified version of the In-Place Text Editor (page 873) that consists of a bounding box that is the height of the text and expands as you type. Right-click to select options on the shortcut menu.

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for paper height and rotation angle. The text that you enter in the In-Place Text Editor for single-line text is placed directly beneath the previous line of text. The point that you specified at the prompt is also stored as the insertion point of the text.

If the *DTEXTED* system variable is set to 1, text created using TEXT or DTEXT displays the Edit Text dialog box. (page 382)

If *DTEXTED* is set to 2, the In-Place Text Editor is displayed. When creating text, you can click anywhere in a drawing to create a new text block. You can also use the keyboard to move among text blocks (for example: for new text created using the *TEXT* command, you can navigate through text groups by

pressing TAB or Shift+TAB, or edit a group of text lines by pressing ALT and clicking each text object.)

**NOTE** Text that would otherwise be difficult to read (if it is very small, very large, or is rotated) is displayed at a legible size and is oriented horizontally so that you can easily read and edit it.

You can enter special characters and format text by entering Unicode strings (page 1431) and control codes (page 1431).

Use -TEXT to honor the TEXTEVAL system variable (page 1432).

DTEXT is the same as TEXT.

### Start Point

Specifies a start point for the text object.

Specify height <*current*>: *Specify a point (1), enter a value, or press* ENTER

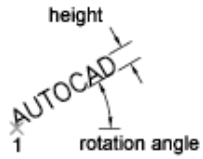
The SpecifyHeight prompt is displayed only if the current text style is not and does not have a fixed height.

Specify paper text height <*current*>: *Specify a height, or press* ENTER

The Specify Paper Text Height prompt is displayed only if the current text style is annotative.

Specify rotation angle of text <*current*>: *Specify an angle or press* ENTER

Enter text in the In-Place Text Editor for single-line text.




### Justify

Controls justification of the text.

Enter an option
[Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

You can also enter any of these options at the Specify Start Point of Text prompt.

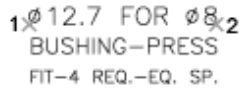
**Align** Specifies both text height and text orientation by designating the endpoints of the baseline.

Specify first endpoint of text baseline: *Specify a point (1)*

Specify second endpoint of text baseline: *Specify a point (2)*

Enter text in the In-Place Text Editor for single-line text.

The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.



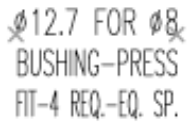

**Fit** Specifies that text fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.

Specify first endpoint of text baseline: *Specify a point (1)*

Specify second endpoint of text baseline: *Specify a point (2)*

Specify height <*current*>:

Enter text in the In-Place Text Editor for single-line text.




The height is the distance in drawing units that the uppercase letters extend from the baseline. Designated text height is the distance between the start point and a point you specify. The longer the text string, the narrower the characters. The height of the characters remains constant.

**Center** Aligns text from the horizontal center of the baseline, which you specify with a point.

Specify center point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.

The rotation angle specifies the orientation of the text baseline with respect to the center point. You can designate the angle by specifying a point. The text baseline runs from the start point toward the specified point. If you specify a point to the left of the center point, the text is drawn upside down.


**Middle** Aligns text at the horizontal center of the baseline and the vertical center of the height you specify. Middle-aligned text does not rest on the baseline.

Specify middle point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.

The Middle option differs from the MC option in that it uses the midpoint of all text, including descenders. The MC option uses the midpoint of the height of uppercase letters.


**Right** Right-justifies the text at the baseline, which you specify with a point.

Specify right endpoint of text baseline: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**TL (Top Left)** Left-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-left point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**TC (Top Center)** Centers text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-center point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**TR (Top Right)** Right-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-right point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**ML (Middle Left)** Left-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-left point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**MC (Middle Center)** Centers the text both horizontally and vertically at the middle of the text. Available for horizontally oriented text only.

Specify middle-center point of text: *Specify a point (1)*

Specify height of text <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.

The MC option differs from the Middle option in that it uses the midpoint of the height of uppercase letters. The Middle option uses the midpoint of all text, including descenders.

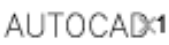

**MR (Middle Right)** Right-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-right point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.




**BL (Bottom Left)** Left-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-left point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**BC (Bottom Center)** Centers text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-center point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.


**BR (Bottom Right)** Right-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-right point of text: *Specify a point (1)*

Specify height <*current*>:

Specify rotation angle of text <*current*>:

Enter text in the In-Place Text Editor for single-line text.

#### Style

Specifies the text style, which determines the appearance of the text characters. Text you create uses the current text style.

Enter style name or [?] <*current*>: *Enter a text style name or enter* **?** *to list all text styles*

Entering **?** lists the current text styles, associated font files, height, and other parameters.


## Text Shortcut Menu

### Quick Reference

Displays options available for creating and modifying single-line text.

**Opaque Background** When checked, makes the background of the editor opaque. By default, the editor is transparent.

**Insert Field** Displays the Field dialog box (page 578), where you can select a field to insert in the text. When the dialog box closes, the current value of the field is displayed in the text.

**Find and Replace** Displays the Replace dialog box (page 884).

**Select All** Selects all the text in the single-line text object.

**Change Case** Changes the case of selected text. Options are Uppercase and Lowercase.

# Special Unicode Characters

## Quick Reference

When entering text, you can create special characters, including the degree symbol, plus/minus tolerance symbol, and the diameter symbol, by entering the following Unicode character strings:

**\U+00B0** Degrees symbol (°)

**\U+00B1** ±±Tolerance symbol (±)

**\U+2205** Diameter symbol ( ∅ )

See "Unicode Font Descriptions" in the *Customization Guide*.

# Control Codes and Special Characters

## Quick Reference

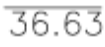
In addition to using Unicode characters for entering special characters, you can also overscore text, underscore text, or insert a special character by including control information in the text string. Use a pair of percent signs to introduce each control sequence.

You can use this control code with standard AutoCAD text fonts and Adobe PostScript fonts:

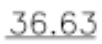
**%%***nnn* Draws character number *nnn*.

You can use these control codes with standard AutoCAD text fonts only:

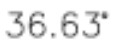
**%%o** Toggles overscoring on and off.




**%%u** Toggles underscoring on and off.

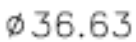



**%%d** Draws degrees symbol (°).



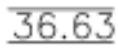

**%%p** Draws plus/minus tolerance symbol (±).


**%%c** Draws circle diameter dimensioning symbol (ý).




**%%%** Draws a single percent sign (%). This is valid for the TEXT command only.


Overscoring and underscoring can be in effect at the same time. Both turn off automatically at the end of the text string.




You can use the %%*nnn* control sequence to display special characters using the PostScript fonts.

A sample drawing *(truetype.dwg)* showing the character map for each font is provided in the *sample* folder.

#### The Euro Symbol

You can use the euro symbol with *.shx* fonts and their TrueType equivalent fonts shipped with AutoCAD 2000 and later releases. If your keyboard does not contain a euro symbol, hold down the ALT key and enter **0128** on the numeric keypad.

## TEXT and the TEXTEVAL System Variable

### Quick Reference

Entering **-text** at the Command prompt displays the same prompts as the TEXT command. Unlike the TEXT command, -TEXT honors the setting of the *TEXTEVAL* system variable. When the TEXTEVAL system variable is set to 2, AutoLISP® expressions are evaluated when the -TEXT command ends. AutoLISP

expressions that are entered using this method must begin with an exclamation point or left parenthesis.

The TEXT command honors the TEXTEVAL system variable setting only if used in a script or AutoLISP expression and all the TEXT command prompts are included within the script or AutoLISP expression.

# TEXTSCR

## Quick Reference

Opens the text window

View ➤ Display ➤ Text WindowAt the Command prompt, enter textscr.
**textscr** (or '**textscr** for transparent use)

TEXTSCR displays the command prompt in a separate window. You can press F2 to toggle between the drawing area and the text window. This command is ignored on dual-screen systems.

When the command prompt is hidden, you can turn it back on by entering **commandline** in the text window.

# TEXTTOFRONT

## Quick Reference

Brings text and dimensions in front of all other objects in the drawing

Tools ➤ Draw Order ➤ Bring Text and Dimensions to Front
**texttofront**
Bring to front: [Text/Dimensions/Both]<Both>: *Enter an option or press* ENTER
**Text** Brings all text in front of all other objects in the drawing.

**Dimensions** Brings all dimensions in front of all other objects in the drawing.

**Both** Brings all text and dimensions in front of all other objects in the drawing.

# THICKEN

## Quick Reference

Creates a 3D solid by thickening a surface

Modify ➤ 3D Operations ➤ ThickenAt the Command prompt, enter Thicken.
**thicken**
3D Make panel (click icon to expand), Thicken Surface
Select surfaces to thicken: *Select one or more surfaces to thicken into solids*
Specify thickness <default>: *Specify a thickness value*

Initially, the default thickness is not set to any value. During a drawing session, the default value for the thickness is always the previously entered thickness value.

The *DELOBJ* system variable controls whether the object(s) you select are automatically deleted when the surface is created or whether you are prompted to delete the object(s).

# TIFOUT

## Quick Reference

Saves selected objects to a file in TIFF file format

**tifout**

The Create Raster File dialog box (a standard file selection dialog box (page 931)) is displayed. Enter the file name in the dialog box.

Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A TIFF file is created that contains the objects you select. The file reflects what is displayed on the screen.

---

**NOTE** When the *FILEDIA* system variable is set to 0 (Off), command prompts are displayed.

---

# TIME

## Quick Reference

Displays the date and time statistics of a drawing

Tools ➤ Inquiry ➤ TimeAt the Command prompt, enter time.
**time** (or '**time** for transparent use)

TIME displays the following information:

Current time (page 1435): Wednesday, December 31, 2003 9:54:51:406 AM
Times for this drawing:
Created (page 1435): Friday, December 12, 2003 1:21:36:203 AM
Last Updated (page 1435): Wednesday, December 31, 2003 9:49:19:208 AM
Total Editing Time (page 1435): 0 days 06:44:10.520
Elapsed Timer (page 1435) (on): 0 days 00:07:05.312
Next Automatic Save In (page 1435): 0 days 01:59:15.570
Enter option [Display (page 1435)/On (page 1435)/OFF (page 1436)/Reset (page 1436)]:
*Enter an option or press* ENTER
**Current Time** Displays the current date and time to the nearest millisecond using a 24-hour clock.

**Created** Displays the date and time that the current drawing was created.

**Last Updated** Displays the date and time of the latest update of the current drawing. This date and time is initially the drawing creation time. The time is revised whenever the drawing file is saved.

**Total Editing Time** Displays the time spent editing the current drawing. This timer is updated by the program and cannot be reset or stopped. Plotting time is not included in the total editing time. If you quit the editing session without saving the drawing, the time you spent in the editing session is not added to the accumulated editing time.

**Elapsed Timer** Runs as another timer while the program is running. You can turn it on and off or reset it whenever you like.

**Next Automatic Save In** Indicates the time remaining until the next automatic save. You can set the time interval using *OPTIONS* or the *SAVETIME* system variable.

**Display** Repeats the display with updated times.

**On** Starts the user elapsed timer if it was off.

**Off** Stops the user elapsed timer.

**Reset** Resets the user elapsed timer to 0 days 00:00:00.000.

# TINSERT

### Quick Reference

Inserts a block in a table cell

With a cell in a table selected, right-click and click Insert Block on the shortcut menu.
**tinsert**

The Insert a Block in a Table Cell dialog box (page 1436) is displayed.

## Insert a Block in a Table Cell Dialog Box

### Quick Reference

With a cell in a table selected, right-click and click Insert Block on the shortcut menu.
**tinsert**

Specifies options for inserting a block in a table cell.

**Scale** Specifies the scale for the block reference. Enter a value or select AutoFit to scale the block to fit in the selected cell.

**Rotation Angle** Specifies a rotation angle for the block.

**Cell Alignment** Specifies alignment for the block in the table cell. The block is middle-, top-, or bottom-aligned with respect to the top and bottom borders of the cell. The block is center-, left-, or right-aligned with respect to the left and right borders of the cell.

# TOLERANCE

## Quick Reference

Creates geometric tolerances

Dimension 
Dimension ➤ ToleranceAt the Command prompt, enter tolerance.
**tolerance**

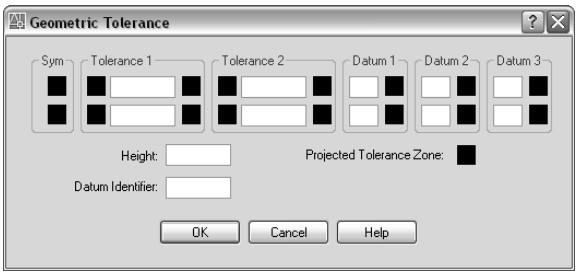
The Geometric Tolerance dialog box (page 1437) is displayed.

# Geometric Tolerance Dialog Box

## Quick Reference

Dimension
Dimension ➤ ToleranceAt the Command prompt, enter tolerance.
**tolerance**

Specifies the symbols and values for a feature control frame.




After you select geometric characteristic symbols, the Geometric Tolerance dialog box closes and the following prompt is displayed:

Enter tolerance location: *Specify a location*

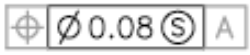
The feature control frame is placed at the specified location.

### Sym

Displays the geometric characteristic symbol, which you select from the Symbol dialog box (page 1440). The dialog box is displayed when you select one of the Sym boxes.


### Tolerance 1

Creates the first tolerance value in the feature control frame. The tolerance value indicates the amount by which the geometric characteristic can deviate from a perfect form. You can insert a diameter symbol before the tolerance value and a material condition symbol after it.




**First Box** Inserts a diameter symbol in front of the tolerance value. Click the box to insert the diameter symbol.

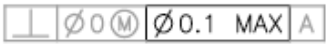
**Second Box** Creates the tolerance value. Enter a value in the box.

**Third Box** Displays the Material Condition dialog box (page 1442), in which you select a modifying symbol. These symbols act as modifiers to the geometric characteristic and the tolerance value of features that can vary in size.

Select the symbol you want to use. The dialog box closes. The symbol is inserted into the MC box for the first tolerance value in the Geometric Tolerance dialog box.

### Tolerance 2

Creates the second tolerance value in the feature control frame. Specify the second tolerance value in the same way as the first.




### Datum 1

Creates the primary datum reference in the feature control frame. The datum reference can consist of a value and a modifying symbol. A datum is a theoretically exact geometric reference used to establish the tolerance zone for a feature.


**First Box** Creates the datum reference value. Enter a value in the box.

**Second Box** Displays the Material Condition dialog box (page 1442), in which you select a modifying symbol. These symbols act as modifiers to the datum reference.

Select the symbol you want to use. The dialog box closes. The symbol is inserted into the MC box for the primary datum reference in the Geometric Tolerance dialog box.

### Datum 2

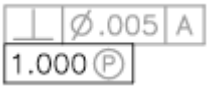
Creates the secondary datum reference in the feature control frame in the same way as the primary datum reference.


### Datum 3

Creates the tertiary datum reference in the feature control frame in the same way as the primary datum reference.

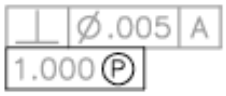

### Height

Creates a projected tolerance zone value in the feature control frame. A projected tolerance zone controls the variation in height of the extended portion of a fixed perpendicular part and refines the tolerance to that specified by positional tolerances. Enter a value in the box.




### Projected Tolerance Zone

Inserts a projected tolerance zone symbol after the projected tolerance zone value.




**Datum Identifier**

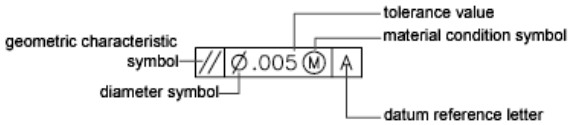
Creates a datum-identifying symbol consisting of a reference letter. A datum is a theoretically exact geometric reference from which you can establish the location and tolerance zones of other features. A point, line, plane, cylinder, or other geometry can serve as a datum. Enter the letter in the box.

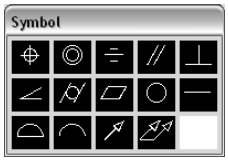

# Symbol Dialog Box

## Quick Reference

Dimension
Dimension ➤ ToleranceAt the Command prompt, enter tolerance.
**tolerance**

Displays the geometric characteristic symbols for location, orientation, form, profile, and runout.



Select the symbol you want to use. The dialog box closes. The symbol is inserted into the Sym text box in the Geometric Tolerance dialog box (page 1437). The following table describes the symbols.

**Geometric characteristic symbols**

| Symbol | Characteristic | Type |
|---|---|---|
|  | Position | Location |
|  | Concentricity or coaxiality | Location |
|  | Symmetry | Location |
|  | Parallelism | Orientation |
|  | Perpendicularity | Orientation |
|  | Angularity | Orientation |
|  | Cylindricity | Form |
|  | Flatness | Form |
|  | Circularity or roundness | Form |
|  | Straightness | Form |

| Geometric characteristic symbols | | |
|---|---|---|
| **Symbol** | **Characteristic** | **Type** |
|  | Profile of a surface | Profile |
|  | Profile of a line | Profile |
|  | Circular runout | Runout |
|  | Total runout | Runout |

# Material Condition Dialog Box

## Quick Reference

Dimension
Dimension ➤ ToleranceAt the Command prompt, enter tolerance.
**tolerance**

Specifies a modifying symbol. These symbols act as modifiers to the geometric characteristic and the tolerance value of features that can vary in size.


Select the symbol you want to use. The dialog box closes. The symbol is inserted into the MC box for the first or second tolerance value in the Geometric Tolerance dialog box (page 1437).

# TOOLBAR

## Quick Reference

Displays, hides, and customizes toolbars

View ➤ ToolbarsAt the Command prompt, enter toolbar.
Right-click any toolbar and choose Customize.
**toolbar**

The Customize User Interface dialog box (page 276) (see *CUI*) is displayed.

If you enter **-toolbar** at the command prompt, options are displayed at the command prompt (page 1443).

# -TOOLBAR

## Quick Reference

If you enter **-toolbar** at the command prompt, the following TOOLBAR command prompts are displayed.

Enter toolbar name (page 1443) or [ALL (page 1445)]: *Enter a name or enter* **all**

### Toolbar Name

Specifies a toolbar to display, close, or position.

Enter a valid toolbar name. If the default menu is loaded, the following names are available:

| | | | |
|---|---|---|---|
| **3d_navigation** | **layouts** | **refedit** | **view** |
| **cad_standards** | **lights** | **reference** | **viewports** |
| **camera_adjustment** | **mapping** | **render** | **visual_styles** |
| **dimension** | **modeling** | **solid_editing** | **walk_and_fly** |
| **draw** | **modify** | **standard** | **web** |

| | | | |
|---|---|---|---|
| **draw_order** | **modify_ii** | **standard_annotation** | **workspaces** |
| **inquiry** | **multileader** | **styles** | **zoom** |
| **insert** | **object_snap** | **text** | |
| **layers** | **orbit** | **ucs** | |
| **layers_ii** | **properties** | **ucs_ii** | |

Enter an option [Show/Hide/Left/Right/Top/Bottom/Float] <Show>: *Enter an option or press* ENTER

**Show** Displays the specified toolbar.

**Hide** Closes the specified toolbar.

**Left** Docks the specified toolbar at the left side of the screen.

Enter new position (horizontal, vertical) <0,0>: *Specify a position or press* ENTER

The Enter New Position prompt sets the position of the toolbar in columns and rows relative to a toolbar dock. The first value is horizontal. The second value is vertical.

**Right** Docks the specified toolbar at the right side of the screen.

Enter new position (horizontal, vertical) <0,0>: *Specify a position or press* ENTER

The Enter New Position prompt sets the position of the toolbar in columns and rows relative to a toolbar dock. The first value is horizontal. The second value is vertical.

**Top** Docks the specified toolbar at the top of the screen.

Enter new position (horizontal, vertical) <0,0>: *Specify a position or press* ENTER

The Enter New Position prompt sets the position of the toolbar in columns and rows relative to a toolbar dock. The first value is horizontal. The second value is vertical.

**Bottom** Docks the specified toolbar at the bottom of the screen.

Enter new position (horizontal, vertical) <0,0>: *Specify a position or press* ENTER

The Enter New Position prompt sets the position of the toolbar in columns and rows relative to a toolbar dock. The first value is horizontal. The second value is vertical.

**Float** Changes the toolbar from docked to floating.

Enter new position (screen coordinates) <0,0>: *Specify a position or press* ENTER

Enter number of rows for toolbar <1>: *Enter a value*

The Enter New Position prompt specifies the location of the floating toolbar in screen coordinate values. The Enter Number of Rows for Toolbar prompt specifies the number of rows in the floating toolbar.

### All

Displays or closes all toolbars.

Enter an option [Show/Hide]: *Enter* **s** *or* **h**
**Show** Displays all toolbars.

**Hide** Closes all toolbars.

# TOOLPALETTES

## Quick Reference

Opens the Tool Palettes window

Standard 
Tools ➤ Palettes ➤ Tool PalettesAt the Command prompt, enter ToolPalettesClose.
**toolpalettes**

Tool palettes organize blocks, hatches and custom tools in an easily available window. The options and settings for tool palettes are accessible from shortcut menus that are displayed when you right-click different areas of the Tool Palettes window. Options on these shortcut menus are listed alphabetically below

---

**NOTE** Tool palettes can be used only in the version of the product in which they were created. For example, you cannot use a tool palette that was created in AutoCAD 2008 in AutoCAD 2007.

---

**Add Text** Inserts a text entry box at the cursor location, where you can add a label that helps organize palette contents.

**Add Separator** Adds a tool palette separator line at the location of the cursor.

**All Palettes** Displays all tool palette tabs in the palette window.

**Allow Docking** Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. A docked window adheres to the side of the application window and causes the drawing area to be resized. Selecting this option also makes Anchor Right and Anchor Left available. Clear this option to undock a docked tool palette.

**Anchor Left/ Anchor Right** Attaches the palette to an anchor tab base at the left or right side of the drawing area. An anchored palette rolls open and closed as the cursor moves across it. When an anchored palette is open, its content overlaps the drawing area. An anchored palette cannot be set to stay open.

**Auto-hide** A setting that causes a floating window to roll open and closed as the cursor moves across it. When this option is cleared, the full tool palette stays open continuously.

**Close** Closes the tool palette window.

**Copy** Copies the selected tool to the Clipboard.

**Customize Commands** Displays the Customize User Interface Dialog Box (page 276).

**Customize Palettes** Displays the Customize dialog box (page 287).

**Cut** Removes the selected tool from the tool palette and places it on the Clipboard.

**Delete** Removes the selected tool from the tool palette.

**Delete Palette** Removes the current palette.

**Dynamic Blocks** Displays the palette tabs containing dynamic blocks.

**Move** Changes the cursor to a four-direction arrow.

**Move Up** Moves the selected tool palette up one position.

**Move Down** Moves the selected tool palette down one position.

**New Palette** Creates a new palette. Enter a name or press ENTER to use the default name.

**Paste** Pastes a tool from the Clipboard to the current tab.

**Properties** Displays the Tool Properties dialog box (page 1448), where you can change the properties of the selected tool.

**Rename** Renames the selected tool.

**Rename Palette** Renames the current palette.

**Samples** Displays the Samples tool palette.

**Sort By** Specifies whether palette contents are sorted by name or by type.

**Size** Changes the cursor to a four-direction arrow. Drag the right edge or the bottom edge of the tool palette to change its size.

**Transparency** Displays the Transparency dialog box (page 994).

**View Options** Displays the View Options dialog box (page 1447), where you can control how tools are displayed.

# View Options Dialog Box

## Quick Reference

Right-click a blank area or a tab on a tool palette and click View Options.

Controls the display of tools in the current tool palette or in all tool palettes.

### Image Size

Controls the display size of the selected tool palette icon.

### View Style

Controls the text displayed with a tool palette icon.

**Icon Only** Displays the tool icon only.

**Icon with Text** Displays the tool icon with the tool name below.

**List View** Displays the tool icon with the tool name to the right.

### Apply To

Controls whether the view options are applied to the current tool palette or to all tool palettes in the Tool Palettes window.

# Tool Properties Dialog Box

## Quick Reference

Right-click a tool on a tool palette and click Properties.

Controls the properties associated with the selected tool. The types of properties that are displayed will vary depending on the type of tool that is selected. In addition the following controls are displayed:

**Image** Displays the icon of the selected tool.

**Name** Displays the name of the selected tool. This can be edited.

**Description** Displays a description of the selected tool.

### Tool Palettes - Materials

When the Tool Palettes - Materials is displayed and a material tool is slected with the right-click Properties is one of the options. By selecting properties the the Tool Properties dialog box displayes the settings from Materials window (page 794).

# Add Actions Dialog Box

## Quick Reference

Adds actions to a parameter or parameter set in the Block Authoring Palettes in the Block Editor (page 160).

**Action Object to Add** Specifies the action to add to the parameter or parameter set.

**Action Object List** Lists the actions associated with the selected parameter or parameter set.

**Add** Adds the action specified in the Action Object to Add box to the parameter or parameter set.

**Delete** Deletes the selected action from the parameter or parameter set.

# TOOLPALETTESCLOSE

## Quick Reference

Closes the Tool Palettes window

Tools ➤ Tool Palettes WindowDoes not exist in the menus.
**toolpalettesclose**

Closes the Tool Palettes window.

# TORUS

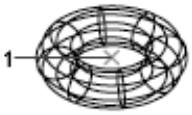
## Quick Reference

Creates a 3D donut-shaped solid

Modeling 
Draw ➤ Modeling ➤ TorusAt the Command prompt, enter torus.
**torus**
3D Make panel, Torus
Specify center point or [3P (page 1450)/2P (page 1450)/TTR (page 1450)]: *Specify a point (1) or or enter an option*

When you specify the center point, the torus is positioned so that its central axis is parallel to the *Z* axis of the current user coordinate system (UCS). The torus is parallel to and bisected by the *XY* plane of the current workplane.

Specify radius (page 1450) or [Diameter (page 1451)] <default>: *Specify a distance or enter* **d**




Initially, the default radius is not set to any value. During a drawing session, the default value for the radius is always the previously entered radius value for any solid primitive.

### 3P (Three Points)

Defines the circumference of the torus with three points that you specify. The three specified points also define the plane of the circumference.

Specify first point: *Specify a point (1)*
Specify second point: *Specify a point (2)*
Specify third point: *Specify a point (3)*

### 2P (Two Points)

Defines the circumference of the torus with two points that you specify. The plane of the circumference is defined by the *Z* value of the first point.

Specify first endpoint of diameter: *Specify a point (1)*
Specify second endpoint of diameter: *Specify a point (2)*

### TTR (Tangent, Tangent, Radius)

Defines the torus with a specified radius tangent to two objects. The specified tangency points are projected onto the current UCS.

Specify point on object for first tangent: *Select a point on an object*
Specify point on object for second tangent: *Select a point on an object*
Specify radius <default>: *Specify a radius or press* ENTER *to specify the default radius value*

Initially, the default radius is not set to any value. During a drawing session, the default value for the radius is always the previously entered radius value for any solid primitive.

### Radius

Defines the radius of the torus: the distance from the center of the torus to the center of the tube. A negative radius creates a solid shaped like an American football.

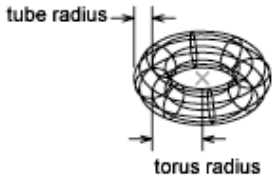
Specify tube radius or [Diameter] <default>: *Specify a distance or enter* **d**

Initially, the default radius is not set to any value. During a drawing session, the default value for the radius is always the previously entered radius value for any solid primitive.

**Radius** Defines the radius of the tube.

**Diameter** Defines the diameter of the tube.

Specify tube diameter: *Specify a nonzero distance*




**Diameter**

Defines the diameter of the torus.

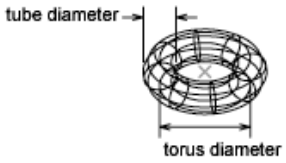
Specify diameter <default>: *Specify a distance*

Initially, the default diameter is not set to any value. During a drawing session, the default value for the diameter is always the previously entered diameter value for any solid primitive.

Specify tube radius or [Diameter]: *Specify a distance or enter* **d**
**Radius** Defines the radius of the tube.

**Diameter** Defines the diameter of the tube.

Specify tube diameter: *Specify a nonzero distance*




# TPNAVIGATE

## Quick Reference

Displays a specified tool palette or palette group

**tpnavigate**
Specify tool palette to display or [palette Group]: *Enter a tool palette name, or enter* **g**
**Tool Palette Name** Displays the specified tool palette.

Specifying a tool palette that is not part of the current group, displays the specified palette and its palette group.

**Palette Group** Displays the following prompt.

Specify palette group to display: *Enter the name of a valid tool palette group.*

Displays the specified tool palette group. The initial prompt repeats so you can specify a tool palette name.

**NOTE** TPNAVIGATE opens the Tool Palettes window if closed.

# TRACE

## Quick Reference

Creates solid lines

**trace**
Specify trace width <*current*>: *Specify a distance or press* ENTER
Specify start point: *Specify a point (1)*
Specify next point: *Specify a point (2)*
Specify next point: *Specify a point (3) or press* ENTER *to end the command*

The endpoints of a trace are on the center line and are always cut square. TRACE automatically calculates the correct bevels for connection to adjacent segments. Each segment is drawn after you either specify the next segment or press ENTER. Because of the way bevels are handled, TRACE has no undo option.

Traces are solid filled when Fill mode is on. When Fill mode is off, only the outline of a trace is displayed.

# TRANSPARENCY

## Quick Reference

Controls whether background pixels in a bitonal image are transparent or opaque

Reference 
Modify ➤ Object ➤ Image ➤ TransparencyAt the Command prompt, enter transparency.

Select an image, right-click in the drawing area, and click Image ➤ Transparency.
**transparency**
Select image(s):
Enter transparency mode [ON/OFF] <*current*>: *Enter an option or press* ENTER
**On** Turns transparency on so that objects beneath the image are visible.

**Off** Turns transparency off so that objects beneath the image are not visible.

---

**NOTE** The *PALETTEOPAQUE* system variable controls whether dockable windows can be made transparent.

---

# TRAYSETTINGS

## Quick Reference

Controls the display of icons and notifications in the status bar tray

Right-click the status bar and click Tray Settings.
**traysettings**

The Tray Settings dialog box (page 1453) is displayed.

# Tray Settings Dialog Box

## Quick Reference

Right-click the status bar, and click Tray Settings.
**traysettings**

Controls the display of icons and notifications in the tray at the right end of the status bar.

### Display Icons from Services

Displays the tray at the right end of the status bar and displays icons from services. When this option is cleared, the tray is not displayed.

#### Display Notifications from Services

Displays notifications from services. When the Display Icons from Services option is cleared, this option is unavailable.

**Display Time** Sets the time in seconds that a notification is displayed.

**Display Until Closed** Displays a notification until you click the Close button.

# TREESTAT

## Quick Reference

Displays information about the drawing's current spatial index

**treestat** (or '**treestat** for transparent use)

The program indexes objects in a region by recording their positions in space. The result is called a *spatial index*. The spatial index is tree structured and has branching nodes to which objects are attached. The index has two major branches. The paper space branch is called a quad-tree and treats objects as two-dimensional. The model space branch is called an oct-tree and treats objects as either two- or three-dimensional. The model space branch can also be changed to a quad-tree when you are working on two-dimensional drawings.

TREESTAT displays information about each branch. The most important information is in the first two lines of the report—number of nodes, number of objects, maximum depth of the branch, and average number of objects per node.

If *REDRAW* and object selection are very slow, you can improve their performance. For example, if there are 50 megabytes of memory available and the current drawing has 50,000 objects with only 1,000 nodes in the index tree, increase the *TREEDEPTH* value to improve performance.

Each node consumes about 80 bytes of memory. The fewer objects per node of the oct-tree, the better the performance.

# TRIM

## Quick Reference

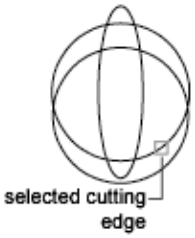
Trims objects at a cutting edge defined by other objects

Modify 

Modify ➤ TrimAt the Command prompt, enter trim.

**trim**

Current settings: Projection = *current,* Edge = *current*
Select cutting edges...
Select objects or <select all>: *Select one or more objects and press* ENTER, *or press* ENTER *to select all displayed objects*




Select the objects that define the cutting edges to which you want to trim an object, or press ENTER to select all displayed objects as potential cutting edges. TRIM projects the cutting edges and the objects to be trimmed onto the *XY* plane of the current user coordinate system (UCS).

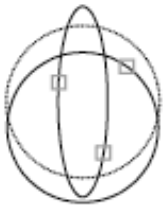
**NOTE** To select cutting edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

Select object to trim (page 1456) or shift-select to extend (page 1456) or [Fence (page 1456)/Crossing (page 1456)/Project (page 1456)/Edge (page 1457)/eRase (page 1458)/Undo (page 1458)]: *Select an object to trim, hold down* SHIFT *and select an object to extend it instead, or enter an option*

Specify an object selection method to select the objects to trim. If more than one trim result is possible, the location of the first selection point determines the result.

### Object to Trim

Specifies the object to trim. The prompt for selecting the object to trim repeats so you can trim multiple objects. Press ENTER to exit the command.




### Shift-Select to Extend

Extends the selected objects rather than trimming them. This option provides an easy method to switch between trimming and extending.

### Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Specify first fence point: *Specify the starting point of the selection fence*
Specify next fence point or [Undo]: *Specify the next point of the selection fence or enter* **u**
Specify next fence point or [Undo]: *Specify the next point of the selection fence, enter* **u**, *or press* ENTER

### Crossing

Selects objects within and crossing a rectangular area defined by two points.

Specify first corner: *Specify a point*
Specify opposite corner: *Specify a point at a diagonal from the first point*

---

**NOTE** Some crossing selections of objects to be trimmed are ambiguous. TRIM resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

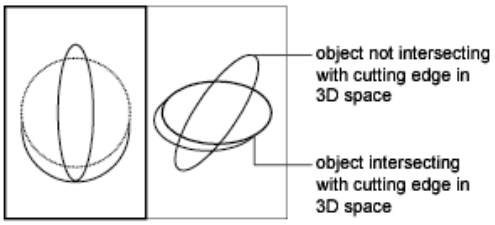
---

### Project

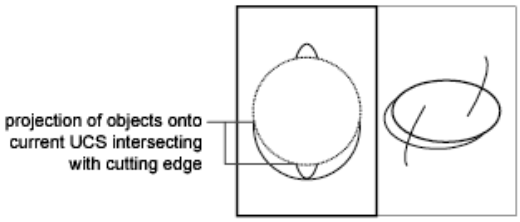
Specifies the projection method used when trimming objects.

Enter a projection option [None/Ucs/View] <*current*>: *Enter an option or press* ENTER

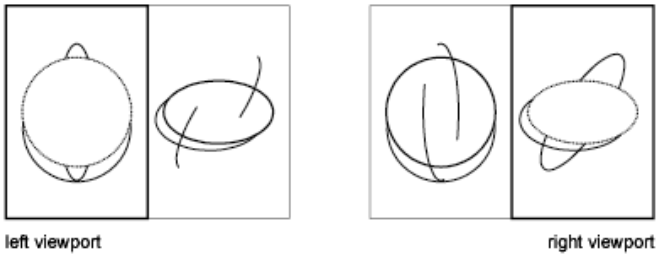
**None** Specifies no projection. The command trims only objects that intersect with the cutting edge in 3D space.




**UCS** Specifies projection onto the *XY* plane of the current UCS. The command trims objects that do not intersect with the cutting edge in 3D space.



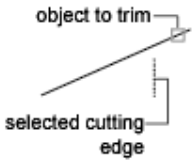

**View** Specifies projection along the current view direction. The command trims objects that intersect the boundary in the current view.



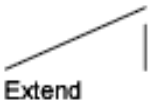

### Edge

Determines whether an object is trimmed at another object's extrapolated edge or only to an object that intersects it in 3D space.

Enter an implied edge extension mode [Extend/No extend] <*current*>: *Enter an option or press* ENTER




**Extend** Extends the cutting edge along its natural path to intersect an object in 3D space.

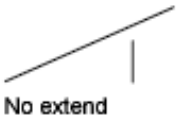



**No Extend** Specifies that the object is trimmed only at a cutting edge that intersects it in 3D space.

---

**NOTE** When trimming hatches, do not set Edge to Extend. If you do, gaps in the trim boundaries will not be bridged when trimming hatches, even when the gap tolerance is set to a correct value.

---




### Erase

Deletes selected objects. This option provides a convenient method to erase unneeded objects without leaving the TRIM command.

Select objects to erase or <exit>: *Use an object selection method and press ENTER to return to the previous prompt*

### Undo

Reverses the most recent change made by TRIM.

# U Commands

# 21

**In this chapter**

- U
- UCS
- UCSICON
- UCSMAN
- UNDEFINE
- UNDO
- UNION
- UNITS
- UPDATEFIELD
- UPDATETHUMBSNOW

# U

## Quick Reference

Reverses the most recent operation

Standard 
With no commands active and no objects selected, right-click in the drawing area and click Undo.
**u**

You can enter **u** as many times as you wish, backing up one step at a time, until the drawing is as it was when you began the current editing session.

When an operation cannot be undone, the command name is displayed but no action is performed. Operations external to the current drawing, such as plotting or writing to a file, cannot be undone.

If you changed modes or used transparent commands during a command, their effects are undone, along with the effects of the main command.

You can also press CTRL+Z to undo.

The U command is equivalent to entering **undo 1**.

# UCS

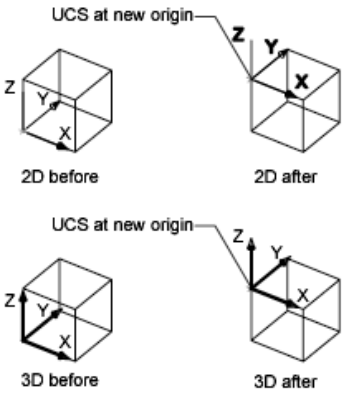
## Quick Reference

Manages user coordinate systems

UCS 
Tools ➤ New UCSDoes not exist in the menus.
**ucs**
Specify Origin of UCS (page 1461) or [Face (page 1462)/NAmed (page 1463)/OBject (page 1464)/Previous (page 1465)/View (page 1466)/World (page 1466)/X/Y/Z (page 1466)/ZAxis (page 1467)] <World>:

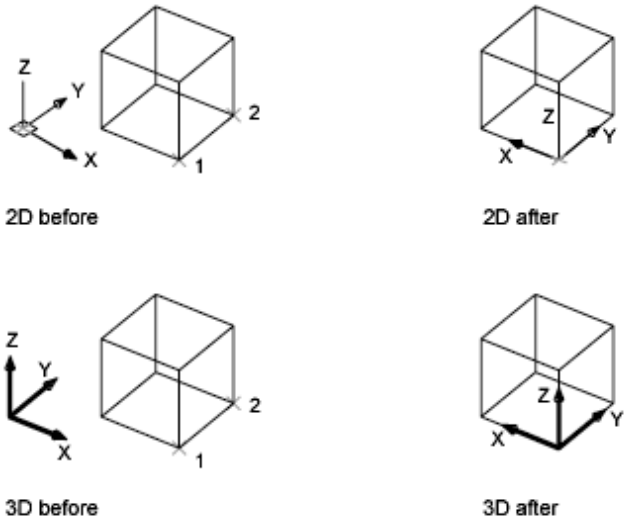
### Specify Origin of UCS

Defines a new UCS using one, two, or three points. If you specify a single point, the origin of the current UCS shifts without changing the orientation of the *X*, *Y*, and *Z* axes.



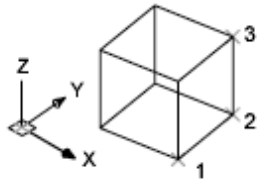

Specify point on X-axis or <Accept>: *Specify a second point or press ENTER to limit input to a single point*

If you specify a second point, the UCS rotates around the previously specified origin point such that the positive *X* axis of the UCS passes through the point.

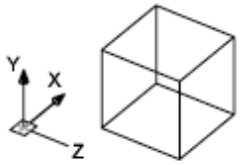


Specify point on XY plane or <Accept>: *Specify a third point or press ENTER to limit input to two points*

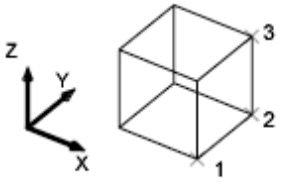
If you specify a third point, the UCS rotates around the *X* axis such that the positive *Y* half of the *XY* plane of the UCS contains the point.



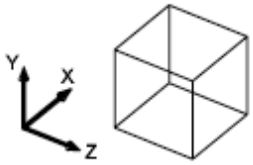

2D before




2D after




3D before




3D after

**NOTE** If you enter the coordinates for a point and you do not specify a *Z* coordinate value, the current *Z* value is used.

### Face

Aligns the UCS to the selected face of a 3D solid. To select a face, click within the boundary of the face or on the edge of the face. The face is highlighted and the UCS *X* axis is aligned with the closest edge of the first face found.

Select face of solid object:
Enter an option [Next/Xflip/Yflip] <accept>:
**Next** Locates the UCS on either the adjacent face or the back face of the selected edge.

**Xflip** Rotates the UCS 180 degrees around the *X* axis.

**Yflip** Rotates the UCS 180 degrees around the *Y* axis.

**Accept** If you press ENTER, accepts the location. The prompt repeats until you accept a location.

### Named

Save and restore commonly used UCS orientations by name.

Enter an option [Restore/Save/Delete/?]: *Specify an option*

### Restore

Restores a saved UCS so that it becomes the current UCS.

Enter name of UCS to restore or [?]: *Enter a name or enter* **?**
**Name** Specifies a named UCS.

**?—List UCSs** Lists the names of currently defined UCSs.

Enter UCS name(s) to list <*>: *Enter a name list or press* ENTER *to list all UCSs*

### Save

Saves the current UCS to a specified name. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used by Microsoft® Windows® and this program for other purposes.

Enter name to save current UCS or [?]: *Enter a name or enter* **?**
**Name** Saves the current UCS with the specified name.

**?—List UCSs** Lists the names of currently defined UCSs.

UCS name(s) to list <*>: *Enter a name list or press* ENTER *to list all UCSs*

### Delete

Removes the specified UCS from the list of saved user coordinate systems.

Enter UCS name(s) to delete <none>: *Enter a name list or press* ENTER

If you delete a named UCS that is current, the current UCS is renamed UNNAMED.

### ?—List UCSs

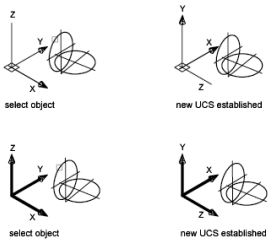
Lists names of user coordinate systems and provides the origin and *X*, *Y*, and *Z* axes for each saved UCS relative to the current UCS. If the current UCS is unnamed, it is listed as WORLD or UNNAMED, depending on whether it's the same as the WCS.

UCS name(s) to list <*>: *Enter a name list*

### Object

Defines a new coordinate system based on a selected 3D object. The new UCS has the same extrusion direction (positive *Z* axis) as that of the selected object.

Select object to align UCS: *Select an object*




The following objects cannot be used with this option: 3D polylines, 3D meshes, and xlines.

For most objects, the origin of the new UCS is located at the nearest vertex to where you select the object, and the X axis is aligned with an edge or is tangent to an edge. For planar objects, the XY plane of the UCS is aligned with the plane in which the object is located. For complex objects, the origin is relocated, but the current orientation of the axes is maintained.

The new UCS is defined as shown in the following table.

**Define a UCS by selecting an object**

| Object | Method of determining UCS |
|---|---|
| Arc | The center of the arc becomes the new UCS origin. The *X* axis passes through the arc endpoint that is closest to the selection point. |
| Circle | The center of the circle becomes the new UCS origin. The *X* axis passes through the selection point. |
| Dimension | The midpoint of the dimension text becomes the new UCS origin. The direction of the new *X* axis is parallel to the *X* axis of the UCS in effect when the dimension was drawn. |

### Define a UCS by selecting an object

| Object | Method of determining UCS |
|---|---|
| Line | The endpoint nearest the selection point becomes the new UCS origin. The new *X* axis is set so that the line lies in the *XZ* plane of the new UCS. The line's second endpoint has a *Y* coordinate of zero in the new UCS. |
| Point | The point becomes the new UCS origin. |
| 2D polyline | The start point of the polyline is the new UCS origin. The *X* axis extends along the line segment from the start point to the next vertex. |
| Solid | The first point of the solid determines the new UCS origin. The new *X* axis lies along the line between the first two points. |
| Trace | The "from" point of the trace becomes the UCS origin, with the *X* axis lying along its centerline. |
| 3D face | The new UCS origin is taken from the first point, the *X* axis from the first two points, and the *Y* positive side from the first and fourth points. The *Z* axis follows by application of the right-hand rule. |
| Shape, text, block reference, attribute definition | The insertion point of the object becomes the new UCS origin, and the new *X* axis is defined by the rotation of the object about its extrusion direction. The object you select to establish a new UCS has a rotation angle of zero in the new UCS. |

### Previous

Restores the previous UCS. The program retains the last 10 coordinate systems created in paper space and the last 10 coordinate systems created in model space. Repeating this option steps back through one set or the other, depending on which space is current.

If you have saved different UCS settings in individual viewports and you switch between viewports, the different UCSs are not retained in the Previous list. However, if you change a UCS setting within a viewport, the last UCS setting

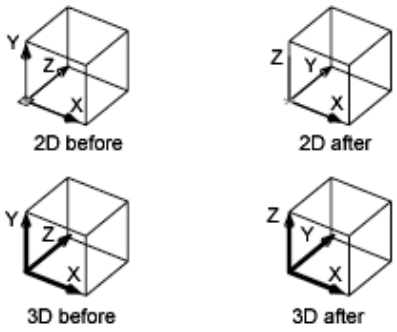
is retained in the Previous list. For example, changing the UCS from World to UCS1 would retain World at the top of the Previous list. If you then switch viewports with Front as the current UCS, and then change the UCS to Right, the Front UCS is retained at the top of the Previous list. If you then choose the UCS Previous option twice in this viewport, the UCS setting changes to Front, and then back to World. See the *UCSVP* system variable.

### View

Establishes a new coordinate system with the *XY* plane perpendicular to your viewing direction (parallel to your screen). The UCS origin remains unchanged.

### World

Sets the current user coordinate system to the world coordinate system. The WCS is the basis for all user coordinate systems and cannot be redefined.

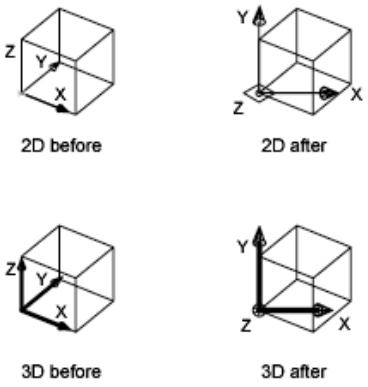



### X,Y, Z

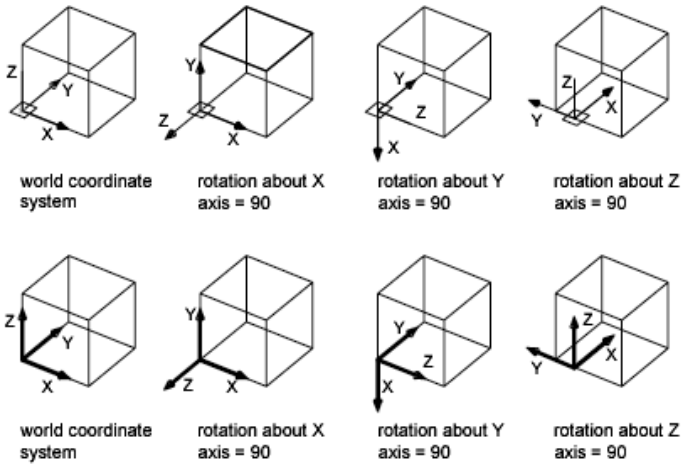
Rotates the current UCS about a specified axis.

Specify rotation angle about *n* axis <0>: *Specify an angle*

In the prompt, *n* is *X*, *Y*, or *Z*. Enter a positive or negative angle to rotate the UCS.



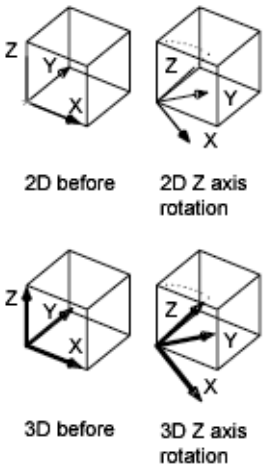

You can define any UCS by specifying an origin and one or more rotations around the *X*, *Y*, or *Z* axis.




### Z Axis

Defines a UCS with a specified positive *Z* axis.

Specify new origin point or [Object] <0,0,0>: *Specify a point or enter* **o**
Specify point on positive portion of Z-axis <*current*>: *Specify a point*




Specify a new origin and a point that lies on the new positive *Z* axis. The *Z* axis option tilts the *XY* plane.

**Object** Aligns the Z axis in the direction tangent to the endpoint nearest to where the object was selected. The positive Z axis points away from the object.

Select object: *Select an open-ended object*

### Apply

Applies the current UCS setting to a specified viewport or all active viewports when other viewports have a different UCS saved in the viewport. The *UCSVP* system variable determines whether a UCS is saved with a viewport.

Pick viewport to apply current UCS or [All] <*current*>: *Specify a viewport by clicking inside it, enter* **a***, or press* ENTER
**Viewport** Applies the current UCS to the specified viewport and ends the UCS command.

**All** Applies the current UCS to all active viewports.

# UCSICON

## Quick Reference

Controls the visibility and placement of the UCS icon

View ➤ Display ➤ UCS IconDoes not exist in the menus.

**ucsicon**

The UCS icon represents the orientation of the user coordinate system (UCS) axes and the location of the current UCS origin. It also represents the current viewing direction relative to the *XY* plane.

A UCS can be stored with a viewport if the *UCSVP* system variable is set to 1 for that viewport.

Different coordinate system icons are displayed in paper space and model space. In both cases, a plus sign (+) appears at the base of the icon when it is positioned at the origin of the current UCS. The letter *W* appears in the *Y* portion of the icon if the current UCS is the same as the world coordinate system.

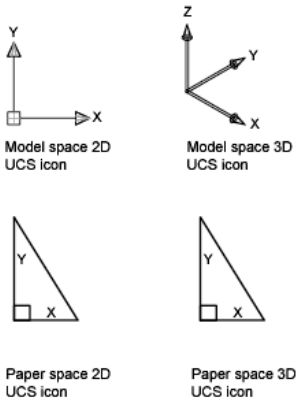
With the 3D UCS icon, a square is displayed in the *XY* plane at the origin when the UCS is the same as the world coordinate system.

A box is formed at the base of the icon if you are viewing the UCS from above (the positive *Z* direction). The box is missing if you are viewing the UCS from below.

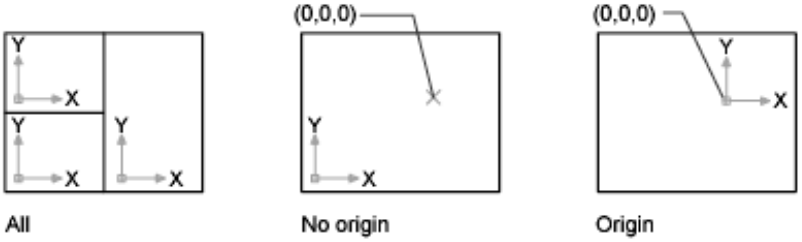
With the 3D UCS icon, the *Z* axis is solid when viewed from above the XY plane and dashed when viewed from below the XY plane.

If the UCS is rotated so that the *Z* axis lies in a plane parallel to the viewing plane—that is, if the *XY* plane is edge-on to the viewer—the 2D UCS icon is replaced by a broken pencil icon.

The 3D UCS icon does not use a broken pencil icon.



Enter an option [ON (page 1470)/OFF (page 1470)/All (page 1470)/Noorigin (page 1470)/ORigin (page 1470)/Properties (page 1470)] <*current*>: *Enter an option or press* ENTER




**On** Displays the UCS icon.

**Off** Turns off display of the UCS icon.

**All** Applies changes to the icon in all active viewports. Otherwise, UCSICON affects only the current viewport.

Enter an option [ON/OFF/Noorigin/ORigin] <*current*>: *Enter an option to apply to all active viewports or press* ENTER

**No Origin** Displays the icon at the lower-left corner of the viewport regardless of the location of the UCS origin.

**Origin** Displays the icon at the origin (0,0,0) of the current coordinate system. If the origin is off the screen, or if the icon cannot be positioned at the origin without being clipped at the viewport edges, the icon is displayed at the lower-left corner of the viewport.

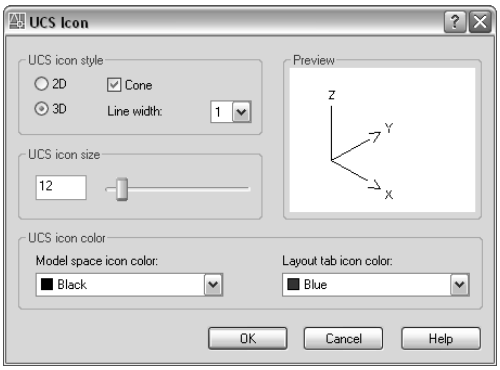
**Properties** Displays the UCS Icon dialog box (page 1470), in which you can control the style, visibility, and location of the UCS icon.

# UCS Icon Dialog Box

## Quick Reference

View ➤ Display ➤ UCS IconDoes not exist in the menus.
**ucsicon**

Controls the display properties of the UCS icon.




### UCS Icon Style

Specifies display of either the 2D or the 3D UCS icon and its appearance.

**2D** Displays a 2D icon without a representation of the *Z* axis.

**3D** Displays a 3D icon.

**Cone** Displays 3D cone arrowheads for the *X* and *Y* axes if the 3D UCS icon is selected. If Cone is cleared, 2D arrowheads are displayed instead.

**Line Width** Controls the line width of the UCS icon if the 3D UCS icon is selected. Available selections are 1, 2, or 3 pixels.

### Preview

Displays a preview of the UCS icon in model space.

### UCS Icon Size

Controls the size of the UCS icon as a percentage of viewport size. The default value is 12, and the valid range is from 5 to 95. Note that the size of the UCS icon is proportional to the size of the viewport in which it is displayed.

### UCS Icon Color

Controls the colors of the UCS icon in model space viewports and in layout tabs.

**Model Space Icon Color** Controls the color of the UCS icon in model space viewports. Click Select Color to open the Select Color dialog box.

You can use the Select Color dialog box to define the color of window elements by selecting from the 255 AutoCAD Color Index (ACI) colors.

**Layout Tab Icon Color** Controls the UCS icon color in layout tabs. Click Select Color to open the Select Color dialog box.

You can use the Select Color dialog box to define the color of window elements by selecting from the 255 AutoCAD Color Index (ACI) colors.

# UCSMAN

## Quick Reference

Manages defined user coordinate systems

UCS

Tools ➤ Named UCSAt the Command prompt, enter ucsman.

**ucsman**

The UCS dialog box (page 1472) is displayed.

# UCS Dialog Box

## Quick Reference

UCS

Tools ➤ Named UCSAt the Command prompt, enter ucsman.

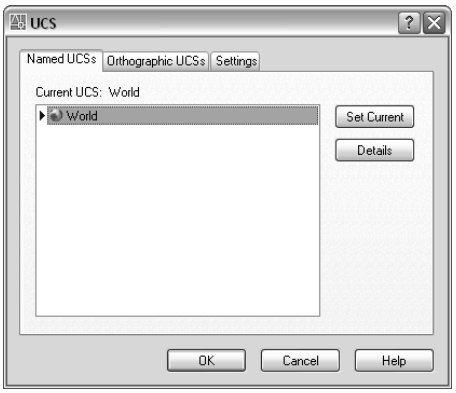
**ucsman**

Displays and modifies defined and unnamed user coordinate systems, restores named and orthographic UCSs, and specifies UCS icon and UCS settings for viewports.

- Named UCSs (page 1472)
- Orthographic UCSs (page 1474)
- Settings (page 1476)

### Named UCSs Tab (UCS Dialog Box)

Lists user coordinate systems and sets the current UCS.




### Current UCS

Displays the name of the current UCS. If the UCS has not been saved and named, it is listed as UNNAMED.

### UCS Names List

Lists the coordinate systems defined in the current drawing. If there are multiple viewports and multiple unnamed UCS settings, the list includes only the unnamed UCS of the current viewport. Unnamed UCS definitions that are locked to other viewports (when the *UCSVP* system variable is set to 1 in that viewport) are not listed in the current viewport. A pointer indicates the current UCS.

UNNAMED is always the first entry if the current UCS is unnamed. World is always listed and cannot be renamed or deleted. If you define other coordinate systems for the active viewport during the current editing session, a Previous entry is next. You can step back through these coordinate systems by selecting Previous and Set Current repeatedly.

To add a UCS name to this list, use the Save option of the *UCS* command. To rename or delete a customized UCS, right-click the UCS name in the list and use the shortcut menu.

### Set Current

Restores the selected coordinate system. You can also restore a selected coordinate system by double-clicking its name in the list, or by right-clicking the name and choosing Set Current. The Current UCS text is updated.

### Details

Displays the UCS Details dialog box (page 1477), which displays UCS coordinate data. You can also view the details about a selected coordinate system by right-clicking the name and choosing Details.

### Rename (Shortcut Menu Only)

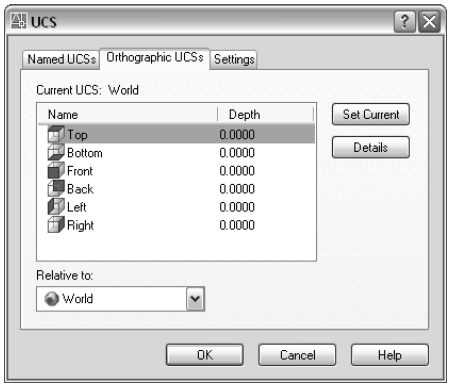
Renames a customized UCS. You cannot rename the World UCS. You can also rename a UCS by double-clicking its name in the list.

### Delete (Shortcut Menu Only)

Deletes a customized UCS. You cannot delete the World UCS.

### Orthographic UCSs Tab (UCS Dialog Box)

Changes the UCS to one of the orthographic UCS settings.




### Current UCS

Displays the name of the current UCS. If the UCS has not been saved and named, it is listed as UNNAMED.

### Orthographic UCS Names

Lists the six orthographic coordinate systems defined in the current drawing. The orthographic coordinate systems are defined relative to the UCS specified in the Relative To list. The Depth column lists the distance between the orthographic coordinate system and the parallel plane passing through the origin of the UCS base setting (stored in the UCSBASE system variable).

**Name** Specifies the name of the orthographic coordinate system.

**Depth** Specifies the distance between the *XY* plane of the orthographic UCS and a parallel plane that passes through the origin of the coordinate system specified by the *UCSBASE* system variable. The parallel plane of the UCSBASE coordinate system can be an *XY*, *YZ*, or *XZ* plane.

---

**NOTE** You can specify the depth or a new origin for the selected orthographic UCS. See Depth option.

---

### Set Current

Restores the selected coordinate system. You can also restore a selected coordinate system by double-clicking its name in the list, or by right-clicking the name and choosing Set Current.

### Details

Displays the UCS Details dialog box (page 1477), which displays UCS coordinate data. You can also view the details about a selected coordinate system by right-clicking the name and choosing Details.

### Relative To

Specifies the base coordinate system for defining the orthographic UCSs. By default, WCS is the base coordinate system. The list displays all the named UCSs in the current drawing.

Whenever you change the Relative To setting, the origin of the selected orthographic UCS is restored to its default position.

If you save an orthographic coordinate system in a drawing as part of a viewport configuration, or if you select a setting in Relative To other than World, the orthographic coordinate system name changes to UNNAMED to distinguish it from the predefined orthographic coordinate system.

### Reset (Shortcut Menu Only)

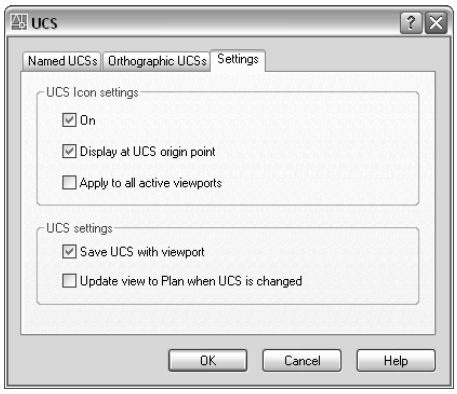
Restores the origin of the selected orthographic coordinate system. The origin can be changed from its default 0,0,0 position using the Move option of the *UCS* command. To reset the origin of the selected orthographic coordinate system, right-click the name in the list and choose Reset Origin from the shortcut menu. The origin is restored to its default location (0,0,0) relative to the specified base coordinate system.

### Depth (Shortcut Menu or Double-Click)

Specifies the distance between the *XY* plane of the orthographic UCS and a parallel plane that passes through the origin of the coordinate system. In the Orthographic UCS Depth dialog box, enter a value or choose the Select New Origin button to use the pointing device to specify a new depth or a new origin.

### Settings Tab (UCS Dialog Box)

Displays and modifies UCS icon settings and UCS settings saved with a viewport.




### UCS Icon Settings

Specifies the UCS icon display settings for the current viewport.

**On** Displays the UCS icon in the current viewport.

**Display at UCS Origin Point** Displays the UCS icon at the origin of the current coordinate system in the current viewport. If this option is cleared, or if the origin of the coordinate system is not visible in the viewport, the UCS icon is displayed at the lower-left corner of the viewport.

**Apply to All Active Viewports** Applies the UCS icon settings to all active viewports in the current drawing.

### UCS Settings

Specifies UCS behavior when the UCS setting is updated.

**Save UCS with Viewport** Saves the coordinate system setting with the viewport. This option sets the *UCSVP* system variable. If this option is cleared, the viewport reflects the UCS of the viewport that is current.

**Update View to Plan When UCS Is Changed** Restores Plan view when the coordinate system in the viewport is changed. (*UCSFOLLOW* system variable)

# Orthographic UCS Depth Dialog Box

## Quick Reference

Right-click in the Orthographic UCSs list. Click Depth.

Specifies the depth for an orthographic UCS.

**Front Depth** Specifies the distance between the *XY* plane of the orthographic UCS and a parallel plane that passes through the origin of the coordinate system. Enter a value or click the Select New Origin button to use the pointing device to specify a new depth or a new origin.

**Select New Origin** Temporarily closes the dialog box so that you can use the pointing device to specify a new depth location in the drawing. To reset the origin of the selected orthographic UCSs to the default location (0,0,0), right-click, and click Reset.

# UCS Details Dialog Box

## Quick Reference

UCS
Tools ➤ Named UCSDoes not exist in the menus
**ucsman**

Displays information about the origin and axes of the selected UCS.

By default, the origin and the values for the *X*, *Y*, and *Z* axes are calculated relative to the world coordinate system.

**Name** Displays the name of the current named UCS. By default, this is set to World.

**Origin** Displays the UCS origin relative to the UCS selected in Relative To.

**X Axis** Displays the values for the *X* axis relative to the UCS selected in Relative To.

**Y Axis** Displays the values for the *Y* axis relative to the UCS selected in Relative To.

**Z Axis** Displays the values for the *Z* axis relative to the UCS selected in Relative To.

**Relative To** Sets a base coordinate system for calculating the values for Origin, X Axis, Y Axis, and Z Axis. This option resets the *UCSBASE* system variable.

# UNDEFINE

## Quick Reference

Allows an application-defined command to override an internal command

**undefine**
Enter command name:

Enter a command name to suppress that command. The suppressed command name can then be redefined to perform some other function.

You can undefine only built-in AutoCAD commands. You cannot undefine commands defined by AutoLISP®. This includes ObjectARX™ application commands registered via **acedDefun()**. You also cannot undefine external commands and aliases defined in the *acad.pgp* file.

If an AutoLISP or ObjectARX application has redefined a command with the same name as a built-in AutoCAD command, the application-defined command is active.

You can restore an undefined command with *REDEFINE*.

You can always access a built-in AutoCAD command by preceding the command name with a period (.).

ObjectARX application commands that are registered via **acedRegCmd** can be accessed by preceding the command name with a period (.), followed by the command's group name, followed by another period (.). For example, the MTEXT command can be accessed with **.acad_mtext.mtext**.

To determine command names and groups of an ObjectARX application, use the *ARX* command, and choose the Commands option to see a listing of all currently loaded ObjectARX commands and their group names.

# UNDO

## Quick Reference

Reverses the effect of commands

Standard 
**undo**
Enter the number (page 1479) of operations to undo or [Auto (page 1479)/Control (page 1479)/BEgin/End (page 1480)/Mark/Back (page 1481)]: *Enter a positive number, enter an option, or press* ENTER *to undo a single operation*

UNDO displays the command or system variable name at the command prompt to indicate that you have stepped past the point where the command was used.

---

**NOTE** UNDO has no effect on some commands and system variables, including those that open, close, or save a window or a drawing, display information, change the graphics display, regenerate the drawing, or export the drawing in a different format.

---

### Number

Undoes the specified number of preceding operations. The effect is the same as entering **u** multiple times.

### Auto

Groups the actions of a single command, making them reversible by a single *U* command. When the Auto option is on, starting a command groups all actions until you exit that command. You can undo the group of actions as if it were one action.

UNDO Auto is not available if the Control option has turned off or limited the UNDO feature.

Enter UNDO Auto mode [ON/OFF] <*current*>: *Enter* **on** *or* **off**, *or press* ENTER

### Control

Limits or turns off UNDO.

Enter an UNDO control option [All/None/One/Combine] <All>: *Enter an option or press* ENTER
**All** Turns on the full UNDO command.

**None** Turns off the U and UNDO commands and discards any UNDO command information saved earlier in the editing session. The Undo button on the Standard toolbar is unavailable.

The Auto, Begin, and Mark options are not available when None or One is in effect. If you attempt to use UNDO while it is turned off, the following prompt is displayed:

Enter an UNDO control option [All/None/One] <All>:

**One** Limits UNDO to a single operation.

The Auto, Begin, and Mark options are not available when None or One is in effect. The main prompt for the UNDO command changes to show that only a Control option or a single step of the UNDO command is available when the One option is in effect.

Control/<1>:

If you enter **c**, the previous prompt is displayed:

Enter an UNDO control option [All/None/One] <All>:

**Combine** Controls whether multiple, consecutive zoom and pan commands are combined as a single operation for undo and redo operations.

Combine zoom and pan operations? [Yes/No] <Yes>: *Enter* **y** *or* **n** *or press* ENTER

---

**NOTE** Pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

---

### Begin, End

Groups a sequence of actions into a set. After you enter the Begin option, all subsequent actions become part of this set until you use the End option. Entering **undo begin** while a group is already active ends the current set and begins a new one. UNDO and U treat grouped actions as a single action.

If you enter **undo begin** without **undo end**, using the Number option undoes the specified number of commands but does not back up past the begin point. If you want to go back to before the begin point, you must use the End option, even if the set is empty. The same applies to the U command. A mark placed by the Mark option disappears inside an UNDO group.

**Mark, Back**

Mark places a mark in the undo information. Back undoes all the work done back to this mark. If you undo one operation at a time, you are informed when you reach the mark.

You can place as many marks as necessary. Back moves back one mark at a time, removing the mark. If no mark is found, Back displays the following prompt:

This will undo everything. OK? <Y>: *Enter* **y** *or* **n** *or press* ENTER

Enter **y** to undo all commands entered in the current session. Enter **n** to ignore the Back option.

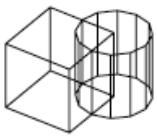
When you use the Number option to undo multiple actions, UNDO stops if it encounters a mark.

# UNION

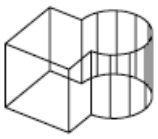
## Quick Reference

Combines selected regions or solids by addition

Modeling 
Modify ➤ Solid Editing ➤ UnionDoes not exist in the menus.
**union**
3D Make panel, Union
Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*



solids before UNION



solids after UNION

The selection set can contain regions and solids that lie in any number of arbitrary planes. The selection sets are divided into subsets that are joined separately. Solids are grouped in the first subset. The first selected region and all subsequent coplanar regions are grouped in the second set. The next region

that is not coplanar with the first region and all subsequent coplanar regions are grouped in the third set, and so on until all regions belong to a subset.


The resulting composite solid includes the volume enclosed by all of the selected solids. Each of the resulting composite regions encloses the area of all regions in a subset.

# UNITS

## Quick Reference

Controls coordinate and angle display formats and precision

Format ➤ UnitsAt the Command prompt, enter units.
**units** (or '**units** for transparent use)

The Drawing Units dialog box (page 1482) is displayed.

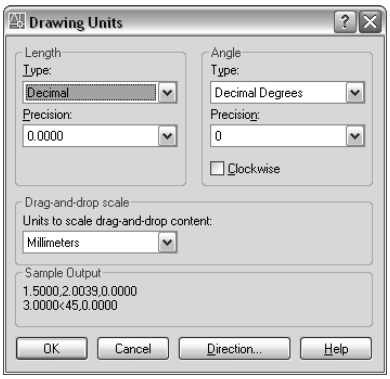
If you enter **-units** at the command prompt, options are displayed at the command prompt (page 1486).

# Drawing Units Dialog Box

## Quick Reference

Format ➤ UnitsAt the Command prompt, enter units.
**units** (or '**units** for transparent use)

Defines the unit and angle formats.




### Length

Specifies the current unit of measurement and the precision for the current units.

**Type** Sets the current format for units of measure. The values include Architectural, Decimal, Engineering, Fractional, and Scientific. The Engineering and Architectural formats produce feet-and-inches displays and assume that each drawing unit represents one inch. The other formats can represent any real-world unit.

**Precision** Sets the number of decimal places or fractional size displayed for linear measurements.

### Angle

Specifies the current angle format and the precision for the current angle display.

**Type** Sets the current angle format.

**Precision** Sets the precision for the current angle display.

The following conventions are used for the various angle measures: decimal degrees appear as decimal numbers, grads appear with a lowercase *g* suffix, and radians appear with a lowercase *r* suffix. The degrees/minutes/seconds format uses *d* for degrees, ' for minutes, and " for seconds; for example:

123d45'56.7"

Surveyor's units show angles as bearings, using N or S for north or south, degrees/minutes/seconds for how far east or west the angle is from direct north or south, and E or W for east or west; for example:

N 45d0'0" E

The angle is always less than 90 degrees and is displayed in the degrees/minutes/seconds format. If the angle is precisely north, south, east, or west, only the single letter representing the compass point is displayed.

**Clockwise** Calculates positive angles in the clockwise direction. The default direction for positive angles is counterclockwise.

When prompted for an angle, you can point in the desired direction or enter an angle regardless of the setting specified for Clockwise.

### Insertion Scale

Controls the unit of measurement for blocks and drawings that are inserted into the current drawing. A block or a drawing that is created with units that are different from the units specified with this option is scaled when inserted. The insertion scale is the ratio of the units used in the source block or drawing and the units used in the target drawing. Select Unitless to insert the block without scaling it to match the specified units.

**NOTE** In the Options dialog box, User Preferences tab, the Source Content Units and Target Drawing Units settings are used when, either in the source block or the target drawing, Insertion Scale is set to Unitless.

### Sample Output

Displays an example of the current settings for units and angles.

### Lighting

Controls the unit of measurement for the intesity of photometric lights in the current drawing.

**NOTE** In order to create and use photometric lights, a unit other than Generic must be specified from the list of options. If Insertion Scale is set to Unitless, a warning message is displayed informing you that the rendered output might not be as expected.

### Direction

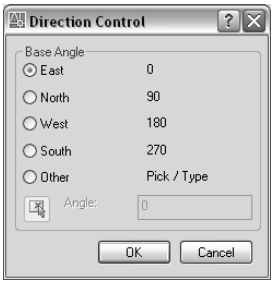
Displays the Direction Control dialog box (page 1485).

# Direction Control Dialog Box

## Quick Reference

Format ➤ UnitsAt the Command prompt, enter units.
**units** (or '**units** for transparent use)

Defines the angle for 0 degrees and specifies the direction in which angles are measured. When prompted for an angle, you can locate a point in the desired direction or enter an angle.




### Base Angle

Sets the direction of the zero angle. The following options affect the entry of angles, the display format, and the entry of polar, cylindrical, and spherical coordinates.

**East** Specifies the compass direction east (the default).

**North** Specifies the compass direction north.

**West** Specifies the compass direction west.

**South** Specifies the compass direction south.

**Other** Specifies a direction different from the points of the compass.

**Angle** Specifies a value for the zero angle when Other is selected. You can specify the angle by entering a value.

**Pick an Angle Button** Defines the zero angle in the graphics area based on the angle of an imaginary line that connects any two points you specify with the pointing device. Available only when Other is selected.

# -UNITS

## Quick Reference

If you enter **-units** at the command prompt, the following UNITS command prompts are displayed.

The text window displays the following prompt:

Report formats: (Examples)
1. Scientific 1.55E+01
2. Decimal 15.50
3. Engineering 1'-3.50"
4. Architectural 1'-3 1/2"
5. Fractional 15 1/2
Enter choice, 1 to 5 <*current*>: *Enter a value (***1**-**5***) or press* ENTER

The following prompt for decimal precision is displayed if you specify the scientific, decimal, or engineering format:

Enter number of digits to right of decimal point (0 to 8) <*current*>: *Enter a value (***0**-**8***) or press* ENTER

The following prompt for the denominator of the smallest fraction is displayed if you specify the architectural or fractional format.

Enter denominator of smallest fraction to display
(1, 2, 4, 8, 16, 32, 64, 128, or 256) <*current*>: *Enter a value (***1**, **2**, **4**, **8**, **16**, **32**, **64**, **128**, *or* **256***) or press* ENTER

The next prompt is for angle formats and precision:

Systems of angle measure: (Examples)
1. Decimal degrees 45.0000
2. Degrees/minutes/seconds 45d0'0"
3. Grads 50.0000g
4. Radians 0.7854r
5. Surveyor's units N 45d0'0" E
Enter choice, 1 to 5 <*current*>: *Enter a value (***1**-**5***) or press* ENTER
Enter number of fractional places for display of angles (0 to 8) <*current*>: *Enter a value (***0**-**8***) or press* ENTER

The next prompt is for the direction for angle 0:

Direction for angle 0:
East 3 o'clock = 0
North 12 o'clock = 90

West 9 o'clock = 180
South 6 o'clock = 270
Enter direction for angle 0 <*current*>: *Enter a value or press* ENTER

The default direction for 0 degrees is to the east quadrant, or 3 o'clock. The default direction for positive angular measurement is counterclockwise.

Measure angles clockwise? [Yes/No] <*current*>: *Enter* **y** *or* **n** *or press* ENTER

# UPDATEFIELD

## Quick Reference

Manually updates fields in selected objects in the drawing

Tools ➤ Update FieldsAt the Command prompt, enter updatefield.
With any text command active and a field selected, right-click and click Update Field.
**updatefield**
Select objects: *Use an object selection method or enter* **all** *to select all fields in the drawing*

Fields in selected objects are updated.

---

**NOTE** The Date field is updated by UPDATEFIELD, but it is not updated automatically based on the setting of the *FIELDEVAL* system variable.

---

# UPDATETHUMBSNOW

## Quick Reference

Manually updates thumbnail previews for sheets, sheet views, and model space views in the Sheet Set Manager

**updatethumbsnow**

Thumbnail previews in the Sheet Set Manager are updated. Sheet thumbnails are displayed on the Sheet List tab, sheet view thumbnails on the View List tab, and model space view thumbnails on the Resource Drawings tab.

# V Commands

# 22

**In this chapter**

- VTOPTIONS

# VBAIDE

## Quick Reference

Displays the Visual Basic Editor

Tools ➤ Macro ➤ Visual Basic EditorAt the Command prompt, enter vbaide.
**vbaide**

The Visual Basic Editor is displayed.

Use the Visual Basic Editor to edit code, forms, and references for any loaded global Visual Basic for Applications (VBA) project or any embedded VBA project in an open drawing. You can also debug and run projects from the Visual Basic Editor.

# VBALOAD

## Quick Reference

Loads a global VBA project into the current work session

Tools ➤ Macro ➤ Load ProjectAt the Command prompt, enter vbaload.
**vbaload**

The Open VBA Project dialog box (a standard file selection dialog box (page 931)) is displayed.

There is no limit to the number of Visual Basic for Applications (VBA) projects you can load. Any project referenced by the selected project will also be loaded, if available.

You cannot use this dialog box to load embedded VBA projects. Embedded projects are loaded when you open the drawing containing the project, and they are unloaded when you close the drawing.

If you select Open Visual Basic Editor, the Visual Basic Editor is opened after the selected project is loaded. For information about the Visual Basic Editor, see the *ActiveX and VBA Developer's Guide*.

If *FILEDIA* is set to 0 (zero), or if you enter **-vbaload** at the command prompt, VBALOAD displays command prompts.

Open VBA project <>: *Enter the path and file name of the project to open*

If the selected project contains macros, the AutoCAD Macro Virus Protection dialog box (page 1492) is displayed.

# AutoCAD Macro Virus Protection Dialog Box

## Quick Reference

Tools ➤ Macro ➤ Load ProjectAt the Command prompt, enter vbaload.
**vbaload**

Disables macro viruses. A macro virus is a type of computer virus that is stored in the macros of a VBA project. If you open a drawing or project file that contains macros, a macro virus can become active and be transmitted to your computer. From that point on, every drawing or project file you save can become infected with the macro virus. When other users open the infected drawing or project file, the macro virus can also be transmitted to their computers.

The AutoCAD Macro Virus Protection dialog box that is displayed when you open a drawing or project file that contains macros gives you the opportunity to disable the macros. A macro virus can be harmful only if it is allowed to run. If you disable the macros, you can open the drawing or project file safely.

The AutoCAD Macro Virus Protection dialog box provides the following options:

**Always Ask Before Opening Projects with Macros** When cleared, prevents the AutoCAD Macro Virus Protection dialog box from being displayed again. You can later enable macro virus protection using the VBA Options dialog box (page 1497), available from the *VBARUN* command.

**Disable Macros** Loads the drawing or project file with the macros disabled. Use Disable Macros if you do not expect the drawing or project file to contain useful macros, or if you aren't certain about the reliability of their source.

Once you disable the macros, you can't run any macros. However, you can still view, edit, and save the macros. You can close the drawing or project file later and open it again with the macros enabled.

**Enable Macros** Loads the drawing or project file with the macros enabled. Use Enable Macros if the drawing or project file is from a reliable source, or if you expect the drawing or project file to contain useful macros.

**Do Not Load** If you are loading a project file, the process is canceled and the project file is not loaded. If you are opening a drawing with an embedded project, the drawing is opened with the macros disabled.

# VBAMAN

## Quick Reference

Loads, unloads, saves, creates, embeds, and extracts VBA projects

Tools ➤ Macro ➤ VBA ManagerAt the Command prompt, enter vbaman.
**vbaman**

The VBA Manager (page 1493) is displayed.

# VBA Manager

## Quick Reference

Tools ➤ Macro ➤ VBA ManagerAt the Command prompt, enter vbaman.
**vbaman**

Loads, unloads, saves, creates, embeds, and extracts Visual Basic for Applications (VBA) projects.

### Drawing

Specifies the active drawing. The list contains all of the drawings that are open in the current work session.

**Embedded Project** Specifies the name of the embedded project for the drawing. If the drawing does not contain an embedded project, "(none)" is displayed.

**Extract** Moves the embedded project out of the drawing and into a global project file. If you have never saved the project, you are prompted to save it.

If you choose Yes, the File Save dialog box is displayed, in which you can specify a file name and location for the project.

If you choose No, the project is extracted and assigned a temporary project name.

If you choose Cancel, the extraction is canceled and you are returned to the VBA Manager.

### Projects

Lists the name and location of all the projects currently available in the current work session.

**Embed** Embeds the selected project in the specified drawing. A drawing can contain only one embedded project. You cannot embed a project in a drawing that already contains an embedded project.

**New** Creates a new project with the default name "Global *n*," where *n* is a session-based number incremented with each new project.

**Save As** Saves a global project. This option is available only when an unsaved global project is selected.

**Load** Displays the Open VBA Project dialog box (see *VBALOAD*), in which you can load an existing project into the current work session.

**Unload** Unloads the selected global project.

**Macros** Displays the Macros dialog box (page 1495), in which you can run, edit, or delete a VBA macro.

**Visual Basic Editor** Displays the Visual Basic Editor in which you can edit code, forms, and references for any loaded global VBA project or any embedded VBA project in an open drawing. You can also debug and run projects from the Visual Basic Editor.

# VBARUN

## Quick Reference

Runs a VBA macro

Tools ➤ Macro ➤ MacrosAt the Command prompt, enter vbarun.
**vbarun**

The Macros dialog box (page 1495) is displayed.

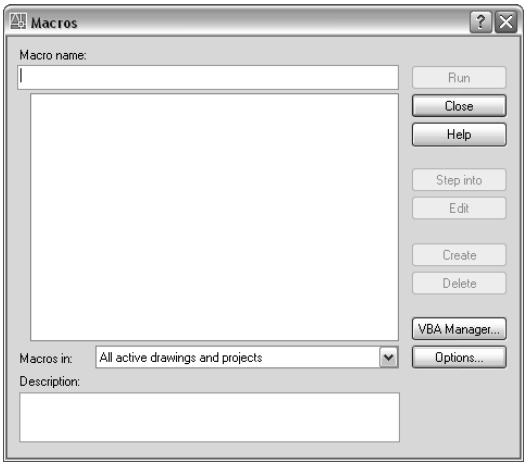
If you enter **-vbarun** at the command prompt, options are displayed at the command prompt (page 1498).

# Macros Dialog Box

## Quick Reference

Tools ➤ Macro ➤ MacrosAt the Command prompt, enter vbarun.
**vbarun**

Runs, edits, or deletes a VBA macro. You can also create new macros, set the VBA options, and display the VBA Manager.




**Macro Name** Specifies the name of the macro you want to run, edit, delete, or create. Select a name from the list of available macros, or enter a name.

**Macro List** Lists all macros found in the drawing or project selected in Macros In. You can double-click a macro name in this list to run the macro.

**Macros In** Specifies the project or drawing whose macros are available from the macro list. You can choose to list the macros in all drawings and projects, all drawings, all projects, any drawing currently open, or any project currently loaded. If your project or drawing is not listed, click VBA Manager to load it.

**Description** Describes the selected macro. You can add or modify the description by directly entering your changes. The new description is saved when you move to a different part of the dialog box.

**Run** Runs the selected macro.

**Step Into** Displays the Visual Basic Editor and begins execution of the macro. Execution is paused at the first executable line of code.

**Edit** Displays the Visual Basic Editor and the selected macro.

**Create** Creates a macro with the name specified in Macro Name, and then displays the Visual Basic Editor and the empty procedure for the new macro.

If no project file or drawing is specified for the new macro, the Select Project dialog box is displayed (page 1496).

If a macro with the specified name already exists, a prompt asks if you want to replace the existing macro.

If you choose Yes, the code in the existing macro is deleted and a new, empty macro is created with the specified name.

If you choose No, you are returned to the Macros dialog box, in which you can enter a new name for the macro.

**Delete** Deletes the selected macro.

**VBA Manager** Displays the VBA Manager (page 1493).

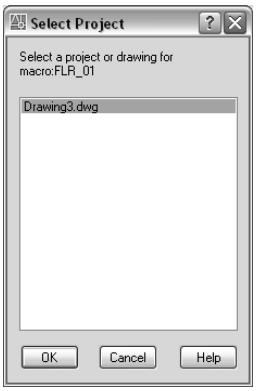
**Options** Displays the VBA Options dialog box (page 1497).

# Select Project Dialog Box

## Quick Reference

Tools ➤ Macro ➤ MacrosAt the Command prompt, enter vbarun.
**vbarun**

Prompts you to select a project or drawing in which to create the new macro. Select a project or drawing from the list and choose OK. The macro is created in the selected project or drawing, and the Visual Basic Editor is opened.




# VBA Options Dialog Box

## Quick Reference

Tools ➤ Macro ➤ MacrosAt the Command prompt, enter vbarun.
**vbarun**

Sets VBA-specific options for the current work session.


**Enable Auto Embedding** Automatically creates an embedded VBA project for all drawings when you open the drawing.

**Allow Break on Errors** Allows VBA to enter Break mode when an error is encountered. Break mode is a temporary suspension of program execution in the development environment. In Break mode, you can examine, debug, reset, step through, or continue program execution.

When you select Allow Break on Errors, unhandled errors found during the execution of a VBA macro suspend the execution of the macro and display the Visual Basic Editor at the point of the error in the macro.

When you clear Allow Break on Errors, untrapped errors found during the execution of a VBA macro display a message alerting you to the error and then end execution of the macro.

**Enable Macro Virus Protection** Enables the virus protection mechanism for VBA macros. The virus protection mechanism displays a built-in warning message whenever you open a drawing that might contain macro viruses.

# -VBARUN

## Quick Reference

If you enter **-vbarun** at the command prompt, the following VBARUN command prompts are displayed.

Macro name: *Enter the name of the macro to run*

For embedded or loaded global projects, enter the name of the macro to run. If the macro name is not unique among all the currently loaded projects, you must also include the name of the project and module in which the macro is found. For example, to run a macro named Test in a project named Project1, and a module named Module1, enter the following information at the Macro name prompt:

**Project1.Module1.Test**

To run a macro in a global project that is not currently loaded, enter the name of the DVB file that contains the macro, as well as the project and module names. For example, if the Test macro described previously is contained in a file called *Acad_Projects.dvb*, and that file is not loaded, you can execute the Test macro by entering the following at the Macro name prompt:

**Acad_Projects.dvb!Project1.Module1.Test**

The *Acad_Projects.dvb* file is loaded and the Test macro is executed. The *Acad_Projects.dvb* file remains loaded once the macro has been completed.

# VBASTMT

## Quick Reference

Executes a VBA statement on the AutoCAD command line

**vbastmt**
Expression: *Enter the VBA statement to execute*

A Visual Basic statement is a complete instruction that can contain keywords, operators, variables, constants, and expressions. A statement generally occupies a single line, although you can use a colon (:) to include more than one statement on a line.

VBA statements are executed in the context of the current drawing.

# VBAUNLOAD

## Quick Reference

Unloads a global VBA project

**vbaunload**
Unload VBA Project: *Enter the name of the project to unload*

If you do not enter a project name, the active global project is unloaded.

# VIEW

## Quick Reference

Saves and restores named views, camera views, layout views, and preset views

View ➤ Named ViewsAt the Command prompt, enter view.
**view**

The View Manager (page 1500) is displayed.

If you enter **-view** at the command prompt, options are displayed at the command prompt (page 1514).

**NOTE** The VIEW command cannot be used transparently.

# View Manager

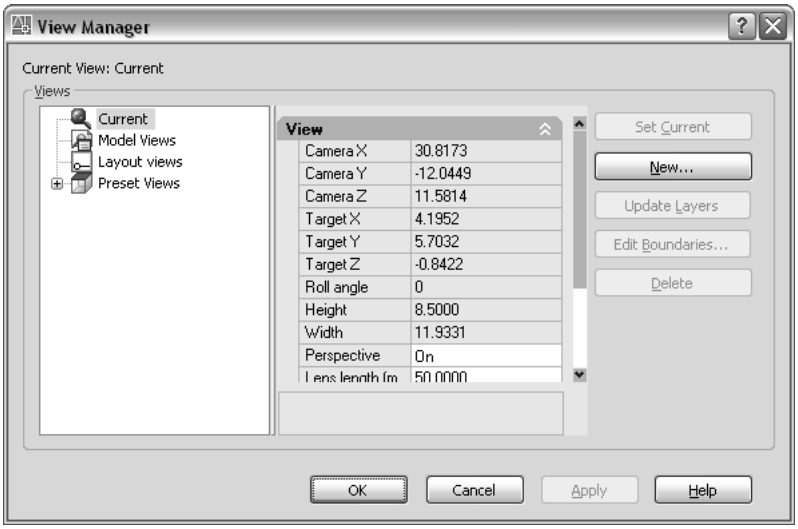
## Quick Reference

View ➤ Named ViewsAt the Command prompt, enter view.
**view**

Creates, sets, renames, modifies, and deletes named views, including model named views and camera views, layout views, and preset views. Click a view to display the properties for that view.

### Views

Displays a list of the available views. You can expand each node (except for the Current node) to display its views.




- **Current**. Displays the current view and its View and Clipping properties
- **Model Views**. Displays a list of named views and cameras, and lists General, View, and Clipping properties for a selected view.

- **Layout Views**. Displays a list of viewports on a layout that define a view, and lists General and View properties for a selected view.
- **Preset Views.** Displays a list of orthogonal and isometric views, and lists the General properties for a selected view.

### General

The following properties are available for views in the General section of the properties list:

**Name** Displays the name of a selected camera, model, or layout view. Cameras are given default names (Camera1, Camera2, and so on) when you create them in a drawing. You can change the name in this field.

**Category** For model views and layout views, displays view categories (for example, *Elevations* or *Section and Details*) defined in a drawing or the sheet set with which the selected view is associated.

**Viewport Association** For layout views only, displays whether the named view is associated with a viewport on a sheet in a sheet set.

**UCS** For model and layout views, displays the name of the user coordinate system (World, Unnamed) saved with the named view.

**Restore Ortho UCS** For preset views, restores the associated UCS when an orthographic view is current. This option sets the *UCSORTHO* system variable.

**Set Relative To** For preset views, specifies the base coordinate system for defining the orthographic view. The *UCSBASE* system variable controls the orientation of orthographic UCSs relative to the WCS or to named UCSs. Changing this setting also changes the orientation of the predefined orthographic UCSs.

**Layer Snapshot** For model views and layout views, specifies whether the current layer on/off settings are stored with the selected view.

**Annotation Scale** Specifies the annotation scale saved with the view.

**Visual Style** For model views only, specifies a visual style to save with the view. If you select None, no visual style will be associated with the view. For more information about visual styles, see Use a Visual Style to Display Your Model.

**Background** For model views whose visual style is not set to 2D Wireframe, specifies the background override (Solid, Gradient, Image, or Sun & Sky) applied to the selected view. Opens the Background dialog box (page 1506).

**Live Section** For model views only, displays the live section applied when the view is restored. For more information about live sections, see *SECTIONPLANE*.

### View

The following view properties are available for views in the View section of the properties list:

**Camera X** For current and model views only, displays the *X* coordinate of the view's camera.

**Camera Y** For current and model views only, displays the *Y* coordinate of the view's camera.

**Camera Z** For current and model views only, displays the *Z* coordinate of the view's camera.

**Target X** For current and model views only, displays the *X* coordinate of the view's target.

**Target Y** For current and model views only, displays the *Y* coordinate of the view's target.

**Target Z** For current and model views only, displays the *Z* coordinate of the view's target.

**Roll angle** Specifies the current roll angle of the view.

**Height** Specifies the height of the view.

**Width** Specifies the width of the view.

**Perspective** For current and model views, specifies whether perspective view is on or off.

**Lens Length (mm)** For all views except layouts, specifies the lens length (in millimeters). Changing this value changes the Field of View setting accordingly.

**Field of View** For all views except layouts, specifies the horizontal field of view (in current angle units). Changing this value changes the Lens Length setting accordingly.

### Clipping

For all views except layouts, the following clipping properties are available for views in the Clipping section of the properties list:

**Front Plane** Specifies the offset value for the front clipping plane if front clipping is enabled for the view.

**Back Plane** Specifies the offset value for the back clipping plane if back clipping is enabled for the view.

**Clipping** Sets clipping options. You can choose Off, Front On, Back On, or Front and Back On.

#### Set Current

Restores the selected view.

#### New

Displays the New View dialog box (page 1503).

#### Update Layers

Updates layer information saved with a selected view to match the layer visibility in the current model space or layout viewport.

#### Edit Boundaries

Displays the selected view, with the rest of the drawing area displayed in a lighter color to show the boundaries of the named view.

#### Delete

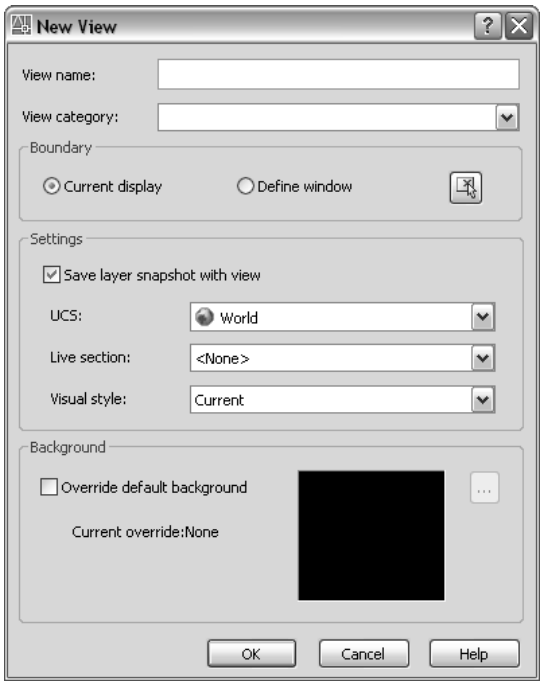
Deletes a selected view.

## New View Dialog Box

### Quick Reference

View ➤ Named ViewsAt the Command prompt, enter view.
**view**

Creates a named view.




### View Name

Specifies the view's name.

### View Category

Specifies a category for the named view. Select a view category from the list, enter a new category, or leave this option empty.

### Current Display

Uses the current display as the new view.

### Define Window

Uses a window as the new view, which you define in the drawing area by specifying two opposite corners.

### Define View Window Button

Temporarily closes the New View and View Manager dialog boxes so that you can use the pointing device to define the opposite corners of the New View window.

### Settings

Provides options for saving settings with the named view.

### Save Layer Snapshot with View

Saves the current layer visibility settings with the new named view.

## UCS

For model and layout views, specifies a UCS to save with the new view.

### Live Section

For model views only, specifies the live section applied when the view is restored.

### Visual Style

For model views only, specifies a visual style to save with the view. For more information about visual styles, see Use a Visual Style to Display Your Model.

### Background Type

For model views whose visual style is not set to 2D Wireframe, specifies the background type (Solid, Gradient, Image, or Sun & Sky) applied to the selected view. Opens the Background dialog box (page 1506) or the Adjust Sun & Sky Background dialog box (page 1508).

**NOTE** The Sun & Sky option is only available when the LIGHTINGUNITS system variable is not set to a value of 0.

#### Save Sun Properties with View

Toggles whether sun and sky data is saved with the named view, the option is automatically selected when choosing Sun & Sky for the background type. Saving sun and sky data to a named view is optional when using a background type other than Sun&Sky.

#### Current Override

Displays the current override type (if one is defined).

**Preview Box** Displays the current background (if one is defined).

#### [...] Button

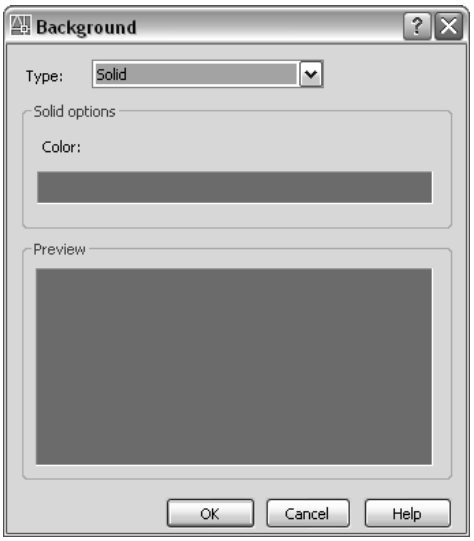
If the Override Default Background option is selected, displays the Background dialog box so you can change the current background selection.

## Background Dialog Box

### Quick Reference

View ➤ Named ViewsAt the Command prompt, enter view.
**view**

Defines the type, color, effects, and position of the background for a view.




### Type

Lists background overrides (None, Solid, Gradient, Image, or Edit) that you can apply to a view.

**Solid** Selects a single-color, solid background.

**Gradient** Specifies a two- or three-color gradient background. (Select the Three Color check box to define a three-color gradient.) Click the color tiles to choose a color. Displays the Select Color dialog box (page 251).

**Image** Uses an image file for the background.

### Solid Options

**Color** Opens the Select Color dialog box (page 251).

**Preview** Displays the selected color.

### Gradient Options

Specifies settings for a new gradient background.

**Three Color** Specifies a three-color gradient. If not selected, you can specify a two-color gradient.

**Top Color** Opens the Select Color dialog box. Choose a color for the top color of the gradient.

**Middle Color** Opens the Select Color dialog box. Choose a color for the middle color of the gradient. This option is displayed only if the Three Color check box is selected.

**Bottom Color** Opens the Select Color dialog box. Choose a color for the bottom color of the gradient.

**Rotation** Specifies an angle at which you can rotate a gradient background. Rotation is not available with a solid or image background. Use the text box or arrows to set the value.

**Preview** Displays the gradient.

#### Image Options

**Filename** Displays the image file's name and path.

**Browse** Displays the Select File dialog box. Select an image file and click Open.

**Adjust Image** Displays the Adjust Background Image dialog box. (page 1512)

#### Preview

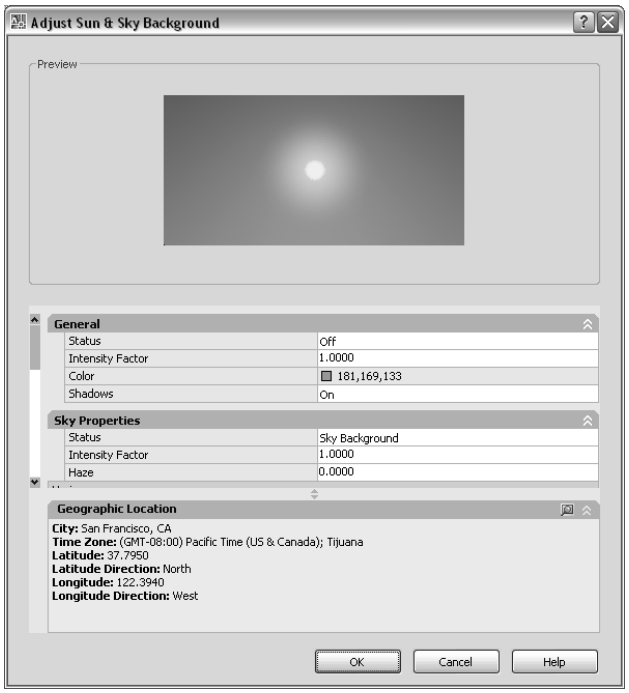
Displays the image.

## Adjust Sun & Sky Background Dialog Box

### Quick Reference

View ➤ Named ViewsAt the Command prompt, enter view.
**view**

Defines the position, color, and effects of the sun and background for a view. Only available in photometric lighting workflow (LIGHTINGUNITS = 1 or 2).




### Preview

Displays the results of the current sun and sky settings for the current view.
.

### General

Sets the general properties of the sun.

**Status** Turns the sun on and off. If lighting is not enabled in the drawing, this setting has no effect.

**Intensity Factor** Sets the intensity or brightness of the sun. The range is from 0 (no light) to maximum. The higher the number, the brighter the light.

**Color** Controls the color of the light. Enter a color name or number or click Select Color to open the Select Color dialog box (page 251).

**Shadows** Turns display and calculation of shadows for the sun on and off. Turning shadows off increases performance.

### Sky Properties

The sky general properties are as follows:

**Status** Determines if the sky illumination is computed at render time. This has no impact on the viewport illumination or the background. It simply makes the sky available as a gathered light source for rendering. This does not control the background. Values are Sky Off, Sky Background, Sky Background and Illumination. [Sky Off] is default.

**Intensity Factor** Provides a way to magnify the effect of the skylight. Values are 0.0-MAX. [1.0] is default.

**Haze** Determines the magnitude of scattering effects in the atmosphere. Values are 0.0-15.0. [0.0] is default.

### Horizon

This category of properties pertains to the appearance and location of the ground plane.

**Height** Determines the absolute position of the ground plane relative to world zero. This parameter represents a world-space length and should be formatted in the current length unit. Values are +-MAX. [0.0] is default.

**Blur** Determines the amount of blurring between ground plane and sky. Values are 0-10. [.1] is default.

**Ground color** Determines the color of the ground plane. Select a color from the drop-down list or select the Select Color dialog box (page 251) to make a color choice.

### Advanced

This category of properties pertains to various artistic effects.

**Night Color** Specifies the color of the night sky. Select a color from the drop-down list or select the Select Color dialog box (page 251) to make a color choice.

**Aerial Perspective** Specifies if aerial perspective is applied. Values are On/Off. [Off] is default.

**Visibility Distance** Specifies the distance at which 10% haze occlusion results. Values are 0.0-MAX. [10000.0] is default.

### Sun Disk Appearance

This category of properties pertains to the background only. They control the appearance of the sun disk.

**Disk Scale** Specifies the scale of the sun disk (1.0 = correct size).

**Glow Intensity** Specifies the intensity of the sun glow. Values are 0.0-25.0. [1.0] is default.

**Disk Intensity** Specifies the intensity of the sun disk. Values are 0.0-25.0. [1.0] is default.

### Sun Angle Calculator

Sets the angle of the sun.

**Date** Displays the current date setting.

**Time** Displays the current time setting.

**Daylight Saving** Displays the current setting for daylight saving time.

**Azimuth** Displays the azimuth, the angle of the sun along the horizon clockwise from due north. This setting is read-only.

**Altitude** Displays the altitude, the angle of the sun vertically from the horizon. The maximum is 90 degrees, or directly overhead. This setting is read-only.

**Source Vector** Displays the coordinates of the source vector, the direction of the sun. This setting is read-only.

### Rendered Shadow Details

Specifies the properties of the shadows.

**Type** Displays the setting for shadow type. This setting is soft (area) only in photometric workflow. This setting is read-only when display of shadows is turned off

**Samples** Specifies the number of samples to take on the solar disk. Values are 0-1000. [8] is default. This setting is read-only when display of shadows is turned off.

**Softness** Displays the setting for the appearance of the edges of shadows. This setting is read-only when display of shadows is turned off.

### Geographic Location

Displays the current geographic location settings. This information is read-only. When a city is not stored with latitude and longitude, the city does not appear in the list.

Use the Edit Geographic Location button to open the Geographic Location dialog box (page 604).

# Adjust Background Image Dialog Box

## Quick Reference

View
View ➤ Named ViewsAt the Command prompt, enter view.
**view**

Controls options for the background image applied to a model space named view.

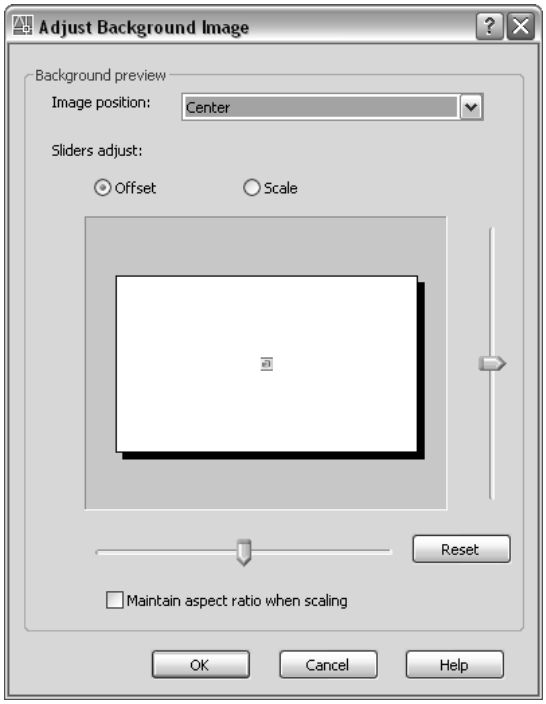


**Image Position** Determines the image position in the named view.

Choose one of the following:

- **Center**. Centers the image without changing its aspect ratio or scale.
- **Stretch**. Centers the image and stretches (scales) it along both the *X* and *Y* axes so that the image takes up the entire view. If you plan to plot the background image, set the image position to Stretch.
- **Tile.** Positions the image at the top left corner of the view and repeats the image as needed to fill up the space in the associated viewport. The image's aspect ratio and scale are maintained.

**Offset** Specifies the image offset control (not available if Stretch is selected as the image position).

Offset values can range from -2000 to +2000 on either the *X* or *Y* axis. Adjust a value by dragging the sliders.

**Scale** Specifies the image scale (not available if Stretch is selected as the image position).

Scale values can range from 0.1 to 10 on either the *X* or *Y* axis. Adjust a value by dragging the sliders.

**Vertical Position Slider** If the Offset option is chosen, offsets the image vertically (the *Y* offset). If Scale is chosen, adjusts the *Y* scale of the image.

**Horizontal Position Slider** If the Offset option is chosen, offsets the image horizontally (the *X* offset). If Scale is chosen, adjusts the *X* scale of the image.

**Reset** Resets the Offset or Scale settings to their default values.

**Maintain Aspect Ratio When Scaling** Locks the *X* and *Y* axes together. Both sliders move accordingly.

If Tile is selected and you change the offset so that the bitmap rectangle appears outside the projection rectangle, the bitmap is not centered within the drawing area when you display the view. (During tiling, the offset acts as a displacement, not an absolute position.) If the bitmap is displayed outside the projection rectangle, it is not displayed in a view.

# -VIEW

## Quick Reference

If you enter **-view** at the command prompt, the following VIEW command prompts are displayed.

Enter an option [? (page 1514)/Delete (page 1514)/Orthographic (page 1514)/Restore (page 1514)/Save (page 1515)/sEttings (page 1515)/Window (page 1516)]:
**?—List Views** Lists the named views and cameras in the drawing.

Enter view names(s) to list <*>: *Enter a wild-card character or press* ENTER

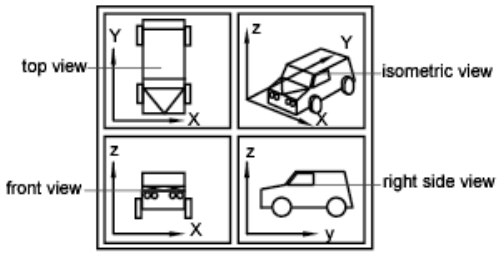
The list includes the name of each specified view and the space in which it was defined. *M* designates model space, and *P* designates paper space.

**Delete** Deletes one or more named views.

Enter view name(s) to delete: *Enter a name or a list of names separated by commas*

**Orthographic** Restores the predefined orthographic view you specify to the current viewport.

Enter an option [Top/Bottom/Front/BAck/Left/Right] <Top>: *Enter an option or press* ENTER




The view orientation of the specified orthographic view is based on the *UCSBASE* system variable, which is set to the world coordinate system by default. When one of the orthographic views is restored, the program zooms to the extents in the current viewport.

**Restore** Restores the view you specify to the current viewport. If a UCS setting was saved with the view, it is also restored.

Enter view name to restore: *Enter a name*

The center point and magnification of the saved view are also restored. If you restore a model space view while working in paper space, you are prompted to select a viewport in which to restore that view.

Restoring model space view.

Select Viewport for view: *Select a viewport*

Select the viewport by clicking its border. The viewport you select must be on and active. The program switches to model space and restores the view in the selected viewport.

If you restore a paper space view while working in model space in a layout tab, the program switches to paper space and restores the view. You can't restore a paper space view if you are working in the Model tab.

**Save** Saves the display in the current viewport using the name you supply.

Enter view name to save: *Enter a name*

The current value of the *UCSVIEW* system variable is displayed when you save a view. To change the setting and turn this option on or off, use the UCS option of VIEW.

**Settings** Specifies various settings for the VIEW command.

Enter an option [Background/Categorize/Layer snapshot/live Section/Ucs/Visual style]:

**Background** Specifies background the for view. Backgrounds are visible in a 3D visual style only.

Enter view name to edit Background or [?]:

Specify the view to update the background for, or enter ? to display a list of all cameras and views in the drawing.

Enter view name(s) to list <*>:

Once the name of a camera or view has been entered, the following prompt is displayed:

Specify background type [Color/Gradient/Image/Sun&sky/None]<None>:

Based on the option entered, either the Background dialog box (page 1506) or the Adjust Sun & Sky Background dialog box (page 1508) is displayed.

**Categorize** Specifies a category for the named view. Named views are displayed by category on the View List tab in the Sheet Set Manager.

**Layer Snapshot** Saves the current layer visibility settings with the new named view.

**Live Section** For model views only, specifies the live section applied when the view is restored.

**UCS** Determines whether the current UCS and elevation settings are saved when a view is saved. Sets the *UCSVIEW* system variable.

**Visual Style** Sets or updates a visual style for a view. Enter a visual style, enter * to specify no visual style, or enter ? to list all visual styles in the drawing.

**Window** Saves a portion of the current display as a view.

Enter view name to save: *Enter a name*

Specify first corner:

Specify opposite corner:

Restoring such a view may display objects outside the window you specified because the shape of the window may differ from that of the viewport in which you are restoring the view. However, plotting the view plots only the objects inside the window, not the entire viewport display.

# VIEWPLOTDETAILS

## Quick Reference

Displays information about completed plot and publish jobs

File ➤ View Plot and Publish Details
**viewplotdetails**

The Plot and Publish Details dialog box (page 1516) is displayed.

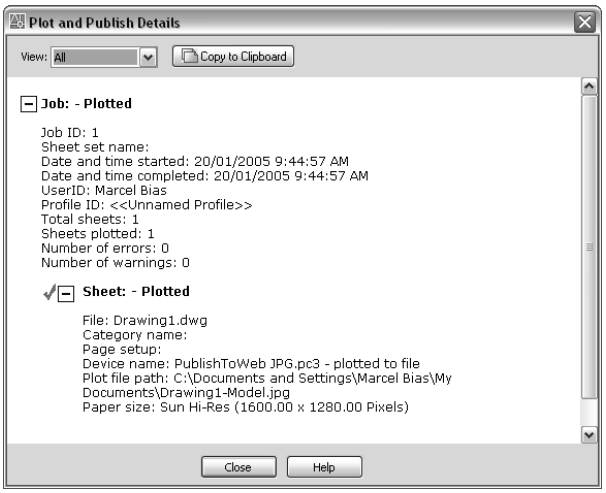
# Plot and Publish Details Dialog Box

## Quick Reference

File ➤ View Plot and Publish Details
**viewplotdetails**

Displays information about plotting and publishing jobs that have been completed in the current session. You can view information about all completed plot and publish jobs or about just the errors that have occurred. You can also copy the information displayed in the dialog box to the Clipboard.

Plot and publish details are also available through the Plot and Publish status bar icon shortcut menu (page 1518).




### View

Specifies what is displayed. You can also right-click in the details area and click View Errors Only or View All to change this setting.

**All** Displays information about all completed plot and publish jobs and sheets within those jobs.

**Errors** Lists errors that have occurred as jobs were plotted or published.

### Copy to Clipboard

Copies all highlighted text in the Plot and Publish Details dialog box to the Clipboard. This text can then be pasted into a text file or other location.

### Details Area

Lists details of completed plot and publish jobs. For each job and sheet, the details area provides the following information:

- Job and sheet names
- Path for jobs plotted to file (if applicable)
- Job completion status, along with whether errors or warnings occurred
- Date and time started and completed
- Total sheets in the job and total sheets plotted

- Number of errors and warnings that occurred
- Detailed information about errors or warning that occurred

# Plot and Publish Status Bar Icon Shortcut Menu

## Quick Reference

When background plotting is enabled, provides options for cancelling the sheet that is currently being plotted, cancelling the entire plot or publish job, viewing the Plot and Publish Details dialog box (page 1516), and viewing the most recently created Design Web Format (DWF™) file. Right-click any of the plot and publish icons in the tray on the right side of the status bar. These icons are described in the following table.

| | |
|---|---|
|  | This animated icon indicates that a plot or publish job is currently being processed in the background |
|  | A plotting details report is available. No errors and warnings occurred. |
|  | A plotting details report is available. Errors and warnings occurred. |

**Cancel Sheet** Cancels the sheet that is currently being plotted.

**Cancel Entire Job** Cancels the entire plot or publish job.

**View Plot and Publish Details** Displays Plot and Publish Details dialog box (page 1516).

**View <filename>.dwf** Opens the most recently created DWF file.

**Enable Balloon Notification** Turns on balloon notification of the status of jobs you have plotted and published.

# VIEWRES

## Quick Reference

Sets the resolution for objects in the current viewport

**viewres**
Do you want fast zooms [Yes/No] <Y>: *Press* ENTER *(Fast Zoom is no longer a functioning option of this command and remains for script compatibility only.)*
Enter circle zoom percent (1-20000) <*current*>: *Enter an integer from 1 to 20,000 or press* ENTER

The model is regenerated.

VIEWRES controls the appearance of circles, arcs, ellipses, and splines using short vectors. The greater the number of vectors, the smoother the appearance of the circle or arc. For example, if you create a very small circle and then zoom in, it might appear to be a polygon. Using VIEWRES to increase the zoom percentage and regenerate the drawing updates and smooths the circle's appearance. Decreasing the zoom percentage has the opposite effect.


**NOTE** Increasing the zoom percentage in VIEWRES may increase the time it takes to regenerate the drawing.

When a paper space layout is made current for the first time and a default viewport is created in the layout, the viewing resolution for this initial viewport is the same as the viewing resolution for the Model tab viewport.

The VIEWRES setting is saved in the drawing. To change the default for new drawings, consider specifying the VIEWRES setting in the template files on which you base your new drawings.

# VISUALSTYLES

### Quick Reference

Creates and modifies visual styles and applies a visual style to a viewport

Tools ➤ Palettes ➤ Visual StylesDoes not exist in the menus.
**visualstyles**
Visual Style panel, Visual Styles Manager

The Visual Styles Manager (page 1520)is displayed.

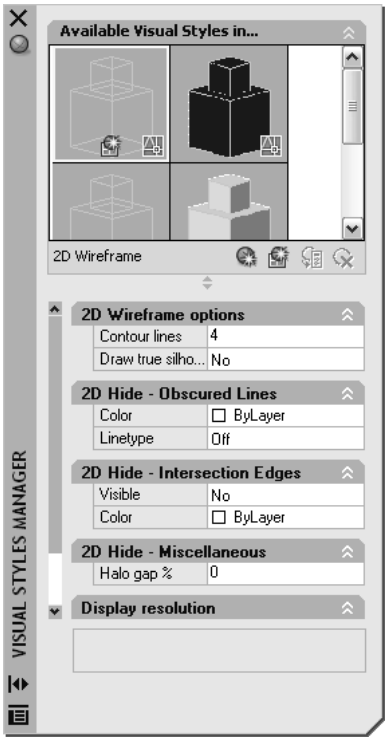
If you enter **-visualstyles** at the command prompt, options are displayed at the command prompt (page 1526).

**NOTE** In a script, the VISUALSTYLES command does not automatically suppress the Visual Styles Manager. You must use **-visualstyles** to display command prompts.

## Visual Styles Manager

### Quick Reference

Tools ➤ Palettes ➤ Visual StylesDoes not exist in the menus.
**visualstyles**
Visual Style panel, Visual Styles Manager




Creates and modifies visual styles.

The Visual Styles Manager has a panel of sample images (page 1521) of the visual styles available in the drawing, and the following properties panels:

- Face settings (page 1523)
- Environment settings (page 1524)
- Edge settings (page 1524)

**NOTE** Changes that you make in the Visual Style panel on the dashboard create a temporary visual style, *Current,* that is applied to the current viewport. The settings are not saved as a named visual style.

### Available Visual Styles in Drawing

Displays sample images of the visual styles that are available in the drawing. The face, environment, and edge settings of the selected visual style are displayed in the settings panels.

A selected visual style displays a yellow border. The name of the selected visual style is displayed at the bottom of the panel.

Icons on the sample image indicate the status of the visual style:

- The icon from the Apply Selected Visual Style to Current Viewport button at center bottom indicates the visual style that is applied to the current viewport.
- A drawing icon at center bottom indicates a visual style that is in use in the current drawing but not in the current viewport.
- A product icon at lower right indicates a default visual style that is shipped with the product.

### Buttons in Tool Strip

Provides button access to frequently used options.

**Create New Visual Style** Displays the Create New Visual Style dialog box, where you enter a name and an optional description. A new sample image is placed at the end of the panel and selected.

**Apply Selected Visual Style to Current Viewport** Applies the selected visual style to the current viewport.

**Export the Selected Visual Style to the Tool Palette** Creates a tool for the selected visual style and places it on the active tool palette. If the Tool Palettes window is closed, it is opened and the tool is placed on the top palette.

**Delete the Selected Visual Style** Removes the visual style from the drawing. A default visual style or one that is in use cannot be removed.

### Shortcut Menu

Provides menu access to options that are available from the buttons in the tool strip as well as the following additional options available only on the shortcut menu. Right-click a sample image in the panel to access the shortcut menu.

**Apply to All Viewports** Applies the selected visual style to all viewports in the drawing.

**Edit Description** Displays the Edit Description dialog box, where you can add a description or change an existing description. The description is displayed in a tooltip when the cursor hovers over the sample image.

**Copy** Copies the visual style sample image to the Clipboard. You can paste it into the Tool Palettes window to create a visual style tool, or you can paste it into the Available Visual Styles panel to create a copy.

**Paste** Pastes a visual style tool into the panel and adds that visual style to the drawing, or pastes a copy of a visual style into the Available Visual Styles panel.

**Size** Sets the size of the sample images. The Full option fills the panel with one image.

**Reset to Default** For one of the default visual styles, restores the original settings.

### Face Settings

Control the appearance of faces in a viewport.

**Highlight Intensity Button** Changes the value for Highlight Intensity from positive to negative and vice versa.

**Opacity Button** Changes the value for Opacity from positive to negative and vice versa.

**Face Style** Defines the shading on faces. (*VSFACESTYLE* system variable)

- Real, the default, is as close as possible to how the face would appear in the real world.
- Gooch uses cool and warm colors instead of dark and light to enhance the display of faces that might be shadowed and difficult to see in a realistic display.
- None does not apply a face style. Other face settings are disabled.

**Lighting Quality** Sets the lighting to show facets on the model or not. Smooth is the default. (*VSLIGHTINGQUALITY* system variable)

**Highlight Intensity** Controls the size of highlights on faces without materials. (*VSFACEHIGHLIGHT* system variable)

**Opacity** Controls the opacity or transparency of faces in a viewport. (*VSFACEOPACITY* system variable)

### Materials and Color

Controls the display of materials and color on faces.

**Materials** Controls whether materials and textures are displayed. (*VSMATERIALMODE* system variable)

**Face Color Mode** Controls the display of colors on faces. (*VSFACECOLORMODE* system variable)

- **Normal**. Does not apply a face color modifier.
- **Monochrome**. Displays the model in shades of the color you specify.
- **Tint**. Changes the hue and saturation value of face colors.
- **Desaturate**. Softens the color by reducing its saturation component by 30 percent.

**Monochrome Color/Tint Color** Displays the Select Color dialog box (page 251), where you select the monochrome or tint color, depending on the face color mode. This setting is not available when face color mode is set to Normal or Desaturate. (*VSMONOCOLOR* system variable)

### Environment Settings

Control shadows and background.

**Shadows** Controls the display of shadows: no shadows, ground shadows only, or full shadows. Turn shadows off to increase performance. (*VSSHADOWS* system variable)

---

**NOTE** To display full shadows, hardware acceleration is required. When Geometry Acceleration is off, full shadows cannot be displayed. (To access these settings, enter **3dconfig** at the Command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.)

---

**Backgrounds** Controls whether backgrounds are displayed in the viewport. (*VSBACKGROUNDS* system variable)

### Edge Settings

Control how edges are displayed.

**Edge Mode** Sets edge display to Facet Edges, Isolines, or None. (*VSEDGES* system variable)

**Color** Displays the Select Color dialog box (page 251), where you can set the color for edges. (*VSEDGECOLOR* system variable)

### Edge Modifiers

Controls settings that apply to all edge modes except None.

**Overhang Button and Setting** Makes lines extend beyond their intersection, for a hand-drawn effect. The button turns the overhang effect on and off. When it is on, you can change the setting. (*VSEDGEOVERHANG* system variable)

**Jitter Button and Setting** Makes lines appear sketched. The settings are Low, Medium, and High; each can be turned off. The button turns the jitter effect on and off. When it is on, you can change the setting. (*VSEDGEJITTER* system variable)

**Crease Angle** Sets the angle at which facet edges within a face are not shown, for a smooth effect. This option is available when the *VSEDGES* system variable is set to display facet edges. (*VSEDGESMOOTH* system variable)

**Halo Gap %** Specifies the size of a gap to be displayed where an object is hidden by another object. This option is available when the Conceptual or 3D Hidden visual styles, or a visual style based on them, is selected. When the halo gap value is greater than 0, sihouette edges are not displayed. (*VSHALOGAP* system variable)

### Fast Silhouette Edges

Controls settings that apply to silhouette edges. Silhouette edges are not displayed on wireframe or transparent objects. (*VSSILHEDGES* system variable)

**Visible** Controls the display of silhouette edges. (*VSSILHEDGES* system variable)

**Width** Specifies the width at which silhouette edges are displayed. (*VSSILHWIDTH* system variable)

### Obscured Edges

Controls settings that apply to obscured edges when edge mode is set to Facet Edges.

**Visible** Controls whether obscured edges are displayed. (*VSOBSCUREDEDGES* system variable)

**Color** Displays the Select Color dialog box (page 251), where you can set the color for obscured edges. (*VSOBSCUREDCOLOR* system variable)

**Linetype** Sets a linetype for obscured edges. (*VSOBSCUREDLTYPE* system variable)

#### Intersection Edges

Controls settings that apply to intersection edges when edge mode is set to Facet Edges.

**Visible** Controls whether intersection edges are displayed. (*VSINTERSECTIONEDGES* system variable)

**NOTE** To increase performance, turn off the display of intersection edges.

**Color** Displays the Select Color dialog box (page 251), where you can set the color for intersection edges. (*VSINTERSECTIONCOLOR* system variable)

**Linetype** Sets a linetype for intersection edges. (*VSINTERSECTIONLTYPE* system variable)

## Create New Visual Style and Edit Name and Description Dialog Boxes

### Quick Reference

Names the visual style and provides space for a description.

**Name** Names the visual style.

**Description** Provides an optional description for the visual style. The description is displayed in a tooltip when the cursor hovers over the sample image.

## -VISUALSTYLES

### Quick Reference

#### visualstyles

If you enter **-visualstyles** at the command prompt, the following VISUALSTYLES command prompts are displayed.

Enter an option [setCurrent (page 1527)/Saveas (page 1527)/Rename (page 1527)/Delete (page 1527)/? (page 1527)]:

### Set Current

Enter an option
[2dwireframe/3dwireframe/3dHidden/Realistic/Conceptual/Other]<*current*>:

These options are the same as the options in *VSCURRENT*.

### Save As

Save current visual style as or [?]: *Enter a name or enter* **?***to list all the visual styles in the drawing*
**Name** Saves the visual style with the name you specify.

---

**NOTE** You must be in model space to save a visual style. If you enter a name that is already in use for a visual style, you can either replace the existing visual style or enter a different name.

---

**?—List Visual Styles** Lists the visual styles in the drawing and redisplays the Save As prompt.

### Rename

Enter visual style to rename [?] <*most recent entry*>:
**Name** Renames the visual style.

**?—List Visual Styles** Lists the visual styles in the drawing and redisplays the Rename prompt.

### Delete

Enter visual style to delete [?]:
**Name** Deletes the visual style.

**?—List Visual Styles** Lists the visual styles in the drawing and redisplays the Delete prompt.

### ?—List Visual Styles

Lists the visual styles in the drawing and redisplays the main prompt.

# VISUALSTYLESCLOSE

## Quick Reference

Closes the Visual Styles Manager

**visualstylesclose**

The Visual Styles Manager (page 1520) closes.

# VLISP

## Quick Reference

Displays the Visual LISP interactive development environment (IDE)

Tools ➤ AutoLISP ➤ Visual LISP EditorAt the Command prompt, enter vlide.
**vlisp**

The Visual LISP IDE is displayed. Use Visual LISP to develop, test, and debug AutoLISP programs.

**NOTE** VLIDE is used to transfer control to Visual LISP. It performs the same function as the VLISP command.

# VPCLIP

## Quick Reference

Clips viewport objects and reshapes the viewport border

Select the viewport to clip, right-click in the drawing area, and choose Viewport Clip.
**vpclip**
Select viewport to clip:
Select clipping object (page 1529) or [Polygonal (page 1529)/Delete (page 1529)] <Polygonal>:

**Clipping Object** Specifies an object to act as a clipping boundary. Objects that are valid as clipping boundaries include closed poly-lines, circles, ellipses, closed splines, and regions.

**Polygonal** Draws a clipping boundary. You can draw line segments or arc segments by specifying points. The following prompt is displayed:

Specify start point:

Specify next point or [Arc/Close/Length/Undo]:

The descriptions of the Next Point, Arc, Close, Length, and Undo options match the descriptions of the corresponding options in the *PLINE* command.

**Delete** Deletes the clipping boundary of a selected viewport. This option is available only if the selected viewport has already been clipped. If you clip a viewport that has been previously clipped, the original clipping boundary is deleted, and the new clipping boundary is applied.

# VPLAYER

## Quick Reference

Sets layer visibility within viewports

**vplayer**
Enter an option [? (page 1529)/Freeze (page 1529)/Thaw (page 1530)/Reset (page 1530)/Newfrz (page 1530)/Vpvisdflt (page 1530)]:

### ?—List Frozen Layers

Displays the names of frozen layers in a selected viewport.

Select a viewport:

The command line window displays the names of any layers that are frozen in the selected viewport.

### Freeze

Freezes a layer or set of layers in one or more viewports. Objects on frozen layers are not displayed, regenerated, or plotted.

Enter layer name(s) to freeze or <select objects>: *Enter one or more layer names*
Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER
**All** Applies the changes in all viewports.

**Select** Temporarily switches to paper space, allowing you to select the viewports where you can apply the layer settings.

Select objects: *Select one or more viewports and press* ENTER

**Current** Applies the changes in the current viewport only.

### Thaw

Thaws layers in specific viewports.

Enter layer name(s) to thaw: *Enter one or more layer names*
Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER
**All** Applies the changes in all viewports.

**Select** Applies the changes in selected viewports.

Select objects: *Select one or more viewports and press* ENTER

**Current** Applies the changes in the current viewport only.

### Reset

Sets the visibility of layers in specified viewports to their current default setting.

Enter layer name(s) to reset: *Enter one or more layer names*
Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER
**All** Applies the changes in all viewports.

**Select** Applies the changes in selected viewports.

Select objects: *Select one or more viewports and press* ENTER

**Current** Applies the changes in the current viewport only.

### Newfrz (New Freeze)

Creates new layers that are frozen in all viewports.

Enter name(s) of new layers frozen in all viewports: *Enter one or more layer names*

### Vpvisdflt (Viewport Visibility Default)

Thaws or freezes the specified layers in subsequently created viewports.

Enter layer name(s) to change viewport visibility: *Enter one or more layer names*
Enter a viewport visibility option [Frozen/Thawed] <*current*>: *Enter* **f** *or* **t***, or press* ENTER

# VPMAX

## Quick Reference

Expands the current layout viewport for editing

Status bar ➤ 
Right-click in a viewport and click Maximize Viewport.
**vpmax**
Select a viewport to maximize:

The viewport is expanded to fill the screen and switched to model space for editing.

# VPMIN

## Quick Reference

Restores the current layout viewport

Status bar ➤ 
Right-click in a maximized viewport and click Restore Viewport.
**vpmin**

The viewport is restored. The center point and magnification are returned to the settings that were in effect before the viewport was maximized.

# VPOINT

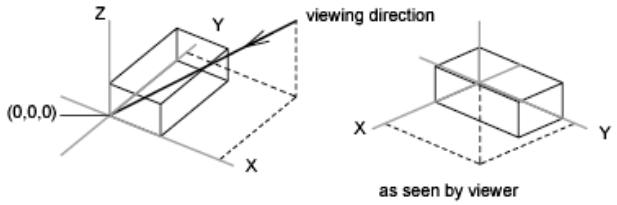
## Quick Reference

Sets the viewing direction for a three-dimensional visualization of the drawing

View ➤ 3D Views ➤ ViewpointAt the Command prompt, enter vpoint.
**vpoint**
Current view direction: VIEWDIR=*current*

Specify a view point (page 1532) or [Rotate (page 1532)] <display compass and tripod (page 1532)>: *Specify a point, enter* **r**, *or press* ENTER *to display a compass and axis tripod*

**View Point** Using the *X*,*Y*,*Z* coordinate you enter, creates a vector that defines a direction from which the drawing can be viewed. The view defined is as if the viewer is looking from the point back at the origin (0,0,0).



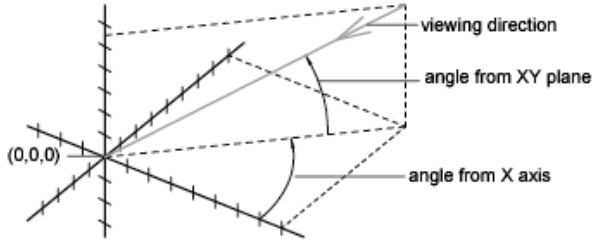

**Rotate** Specifies a new viewing direction using two angles.

Enter angle in XY plane from X axis <*current*>: *Specify an angle*

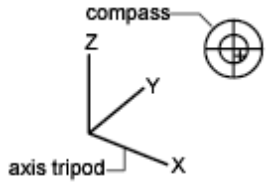
The first angle is specified with respect to the *X* axis, in the *XY* plane.

Enter angle from XY plane <*current*>: *Specify an angle*

The second angle is specified up or down from the *XY* plane.




**Compass and Axis Tripod** Displays a compass and axis tripod, which you use to define a viewing direction in the viewport.




The compass is a two-dimensional representation of a globe. The center point is the north pole (0,0,*n*), the inner ring is the equator (*n*,*n*,0), and the entire outer ring is the south pole (0,0,-*n*).

You can move the small crosshairs on the compass to any portion of the globe with your pointing device. As you move the crosshairs, the axis tripod rotates to conform to the viewing direction indicated on the compass. To select a viewing direction, move your pointing device to a location on the globe and click.

# VPORTS

## Quick Reference

Creates multiple viewports in model space or paper space

Layouts 
View ➤ ViewportsDoes not exist in the menus.
**vports**

The Viewports dialog box (page 1533) is displayed.

If you enter **-vports** at the command prompt, options are displayed at the command prompt (page 1536).

# Viewports Dialog Box

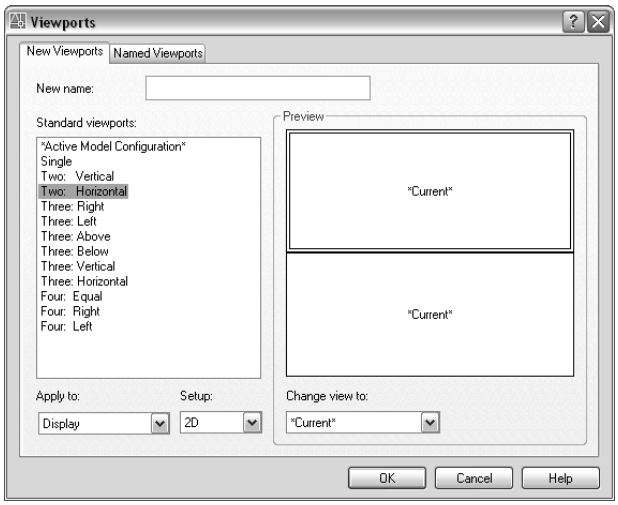
## Quick Reference

Layouts
View ➤ ViewportsDoes not exist in the menus.
**vports**

Creates new viewport configurations, or names and saves a model space viewport configuration. The options available in this dialog box depend on whether you are configuring model space viewports (on the Model tab) or layout viewports (on a layout tab).

### New Viewports Tab—Model Space (Viewports Dialog Box)

Displays a list of standard viewport configurations and configures model space viewports.




**New Name** Specifies a name for the new model space viewport configuration you are creating. If you do not enter a name, the viewport configuration you create is applied but not saved. If a viewport configuration is not saved, it cannot be used in a layout.

**Standard Viewports** Lists and sets the standard viewport configurations, including CURRENT, which is the current configuration.

**Preview** Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

**Apply To** Applies the model space viewport configuration to the entire display or to the current viewport.

- *Display:* Applies the viewport configuration to the entire Model tab display. Display is the default setting.
- *Current Viewport:* Applies the viewport configuration to the current viewport only.

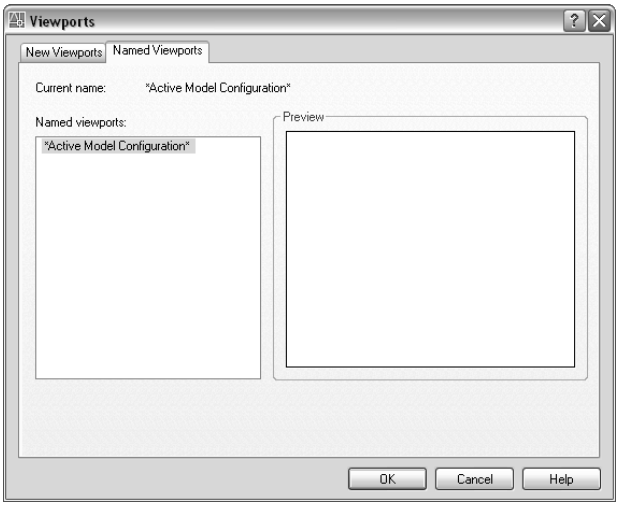
**Setup** Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all of the viewports. When you select 3D, a set of standard orthogonal 3D views is applied to the viewports in the configuration.

**Change View To** Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views. Use the Preview area to see the choices.

**Visual Style** Applies a visual style to the viewport.

### Named Viewports Tab—Model Space (Viewports Dialog Box)

Displays any saved viewport configurations in the drawing. When you select a viewport configuration, the layout of the saved configuration is displayed in Preview.




**Current Name** Displays the name of the current viewport configuration.

### New Viewports Tab—Layouts (Viewports Dialog Box)

Displays a list of standard viewport configurations and configures layout viewports.

**Standard Viewports** Displays a list of standard viewport configurations and configures layout viewports.

**Preview** Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

**Viewport Spacing** Specifies the spacing you want to apply between the layout viewports you are configuring.

**Setup** Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all of the viewports. When you select 3D, a set of standard orthogonal 3D views is applied to the viewports in the configuration.

**Change View To** Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views. Use the Preview area to see the choices.

#### Named Viewports Tab—Layouts (Viewports Dialog Box)

Displays any saved and named model space viewport configurations for you to use in the current layout. You cannot save and name a layout viewport configuration.

# -VPORTS

## Quick Reference

The command prompts available depend on whether you are configuring model viewports (page 1536) (on the Model tab) or layout viewports (page 1538) (on a layout tab).

# -VPORTS - Model Space Viewports

## Quick Reference

If you enter **-vports** at the command prompt from the Model tab, the following VPORTS command prompts are displayed.

Enter an option [Save (page 1536)/Restore (page 1536)/Delete (page 1537)/Join (page 1537)/Single (page 1537)/? (page 1537)/2 (page 1537)/3 (page 1537)/4 (page 1538)] <3>: *Enter an option*

The number and layout of active viewports and their associated settings are called viewport configurations.

**Save** Saves the current viewport configuration using a specified name.

Enter name for new viewport configuration or [?]: *Enter a name or enter* **?** *to list saved viewport configurations*

**Restore** Restores a previously saved viewport configuration.

Enter name of viewport configuration to restore or [?]: *Enter a name or enter* **?** *to list saved viewport configurations*

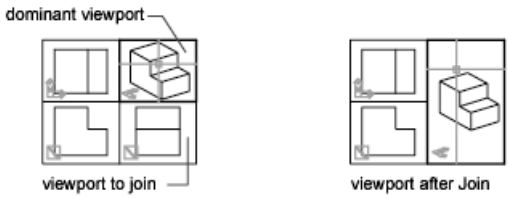
**Delete** Deletes a named viewport configuration.

Enter name(s) of viewport configurations to delete <none>: *Enter a name or enter* **?** *to list saved viewport configurations*

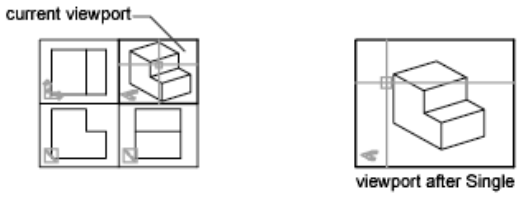
**Join** Combines two adjacent viewports into one larger viewport. The resulting viewport inherits the view of the dominant viewport.

Select dominant viewport <*current*>: *Press* ENTER *or select a viewport*

Select viewport to join: *Select a viewport*




**Single** Returns the drawing to a single viewport view, using the view from the current viewport.




**?—List Viewport Configurations** Displays the identification numbers and screen positions of the active viewports.

Enter name(s) of viewport configuration(s) to list <*>: *Enter a name or press* ENTER

The lower-left and upper-right corners of the viewport define its location. For these corners, values between 0.0,0.0 (for the lower-left corner of the drawing area) and 1.0,1.0 (for the upper-right corner) are used. The current viewport is listed first.

**2** Divides the current viewport in half.

Enter a configuration option [Horizontal/Vertical] <Vertical>: *Enter* **h** *or press* ENTER

**3** Divides the current viewport into three viewports.

Enter a configuration option [Horizontal/Vertical/Above/Below/Left/Right] <Right>: *Enter an option or press* ENTER

The Horizontal and Vertical options split the area into thirds. The Above, Below, Left, and Right options specify where the larger viewport is placed.

**4** Divides the current viewport into four viewports of equal size.

# -VPORTS - Layout Viewports

## Quick Reference

If you enter **-vports** at the command prompt from a layout tab, the following VPORTS command prompts are displayed.

Specify corner of viewport or [ON (page 1538)/OFF (page 1538)/Fit (page 1538)/Shadeplot (page 1538)/Lock (page 1538)/Object (page 1538)/Polygonal (page 1539)/Restore (page 1539)/LAyer (page 1539)/2 (page 1539)/3 (page 1539)/4 (page 1540)] <Fit>: *Specify a point or enter an option*

The number and layout of active viewports and their associated settings are called viewport configurations.

**On** Turns on a viewport, making it active and making its objects visible.

**Off** Turns off a viewport. When a viewport is off, its objects are not displayed, and you cannot make that viewport current.

**Fit** Creates one viewport that fills the available display area. The actual size of the viewport depends on the dimensions of the paper space view.

**Shadeplot** Specifies how viewports in layouts are plotted.

Shade plot? [As displayed/Wireframe/Hidden/Visual styles/Rendered] *<As displayed>*: *Enter a shade plot option*

- *As Displayed:* Plots the same way it is displayed
- *All Visual Styles:* Plots using the specified visual style; all visual styles in the drawing are listed as options whether in use or not
- *All Render Presets:* Plots using the specified render preset; all render presets are listed as options

**Lock** Locks the current viewport. This is similar to layer locking.

**Object** Specifies a closed polyline, ellipse, spline, region, or circle to convert into a viewport. The polyline you specify must be closed and contain at least

three vertices. It can be self-intersecting, and it can contain arcs as well as line segments.

**Polygonal** Creates an irregularly shaped viewport defined by specifying points. For a description of this option, see the New Boundary option under *XCLIP*.

Enter an arc boundary option

[Angle/CEnter/CLose/Direction/Line/Radius/Second pt/Undo/Endpoint of arc] <Endpoint>: *Enter an option*

**Restore** Restores a previously saved viewport configuration.

Enter viewport configuration name or [?]: *Enter a name or enter* **?** *to list saved viewport configurations*

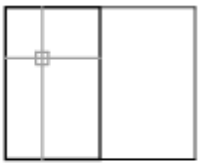
**Layer** Resets layer property overrides for the selected viewport to their global layer properties.

Reset viewport layer property overrides back to global properties [Yes/No]?: *Enter Y to remove all layer property overrides.*

Select viewports: *Select a single or multiple viewports and press* ENTER

**2** Divides the current viewport in half.

Enter viewport arrangement [Horizontal/Vertical]<Vertical>: *Enter* **h** *or press* ENTER

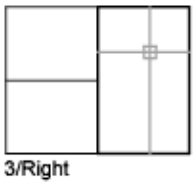


2/Vertical

**3** Divides the current viewport into three viewports.

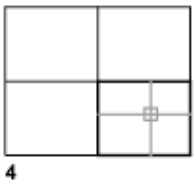
Enter viewport arrangement [Horizontal/Vertical/Above/Below/Left/Right] <Right>: *Enter an option or press* ENTER

Horizontal and Vertical split the area into thirds. The other options create one large viewport in half the available area and two smaller ones in the other half. Above, Below, Left, and Right specify where the larger viewport is placed.



3/Right

**4** Divides the current viewport into four viewports of equal size.




# VSCURRENT

## Quick Reference

Sets the visual style in the current viewport

vscurrent
Enter an option [2dwireframe (page 1540)/3dwireframe (page 1540)/3dHidden (page 1540)/Realistic (page 1540)/Conceptual (page 1540)/Other (page 1540)] <*current*>:

---

**NOTE** To display lighting from point lights, distant lights, spotlights, or the sun, set the visual style to Realistic, Conceptual, or a custom visual style with shaded objects.

---

**2D Wireframe** Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible. Even if the value for the *COMPASS* system variable is set to 1, it does not appear in the 2D Wireframe view.

**3D Wireframe** Displays the objects using lines and curves to represent the boundaries. Displays a shaded 3D UCS icon. You can set the COMPASS system variable to 1 to view the compass.

**3D Hidden** Displays the objects using 3D wireframe representation and hides lines representing back faces.

**Realistic** Shades the objects and smooths the edges between polygon faces. Materials that you have attached to the objects are displayed.

**Conceptual** Shades the objects and smooths the edges between polygon faces. Shading uses a transition between cool and warm colors. The effect is less realistic, but it can make the details of the model easier to see.

**Other** Displays the following prompt:

Enter a visual style name [?]: *Enter the name of a visual style in the current drawing, or enter* **?** *to display a list of names and repeat the prompt*

# VSLIDE

## Quick Reference

Displays an image slide file in the current viewport

**vslide**

The Select Slide File dialog box, a standard file selection dialog box, (page 931) is displayed. Enter a slide file name (*.sld* extension) to display. When you press ENTER or click Open, the slide file is opened.

To display a slide in a slide library (*.slb* extension), set *FILEDIA* to 0, enter **vslide**, and then specify the slide library file name followed by the slide file name enclosed in parentheses: **slidelibrary(slide)**.

When you view slides of images shaded with the -SHADE command in a larger window or a higher resolution than was used for creating the slide, black lines may appear interspersed among the lines of the shaded image. To avoid this situation, use a full screen that is set at the highest resolution when creating the slides.

# VSSAVE

## Quick Reference

Saves a visual style

**vssave**
Save current visual style as or [?]: *Enter a name or enter* **?***to list all the visual styles in the drawing*

---

**NOTE** You must be in model space to save a visual style. If you enter a name that is already in use for a visual style, you can either replace the existing visual style or enter a different name.

---

# VTOPTIONS

## Quick Reference

Displays a change in view as a smooth transition

**vtoptions**

The View Transitions dialog box (page 1542) is displayed.

# View Transitions Dialog Box

## Quick Reference

**vtoptions**

Sets the options for smooth view transitions.

### Enable Animation for Pan and Zoom

Makes a smooth view transition during panning and zooming. (*VTENABLE* system variable)

### Enable Animation When View Rotates

Makes a smooth view transition when the view angle is changed. (VTENABLE system variable)

### Enable Animation During Scripts

Makes smooth view transitions while a script is running. (VTENABLE system variable)

### Transition Speed

Sets the speed of a smooth view transition in milliseconds.

### Performance

Set the minimum speed for a smooth view transition in frames per second. When a smooth view transition cannot maintain this speed, an instant transition is used to preserve performance.

# W Commands

# 23

**In this chapter**

# WALKFLYSETTINGS

## Quick Reference

Specifies walk and fly settings

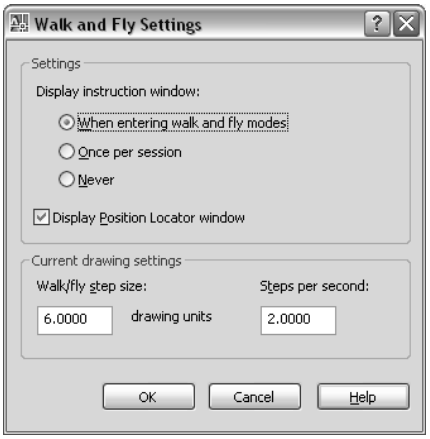
3D Navigation
View ➤ Walk and Fly ➤ Walk and Fly SettingsDoes not appear in the menus.
**walkflysettings**
3D Navigate panel, Walk flyout

The Walk and Fly Settings dialog box (page 1546) is displayed.

# Walk and Fly Settings Dialog Box

## Quick Reference

Specifies walk and fly settings.




**Settings** Specifies settings related to the Instruction window and the Position Locator window (page 54).

**When Entering Walk and Fly Modes** Specifies that the Walk and Fly Navigation Mappings dialog box (page 56) is displayed each time you enter walk or fly mode.

**Once Per Session** Specifies that the Walk and Fly Navigation Mappings dialog box is displayed the first time in each AutoCAD session that you enter walk or fly mode.

**Never** Specifies that the Walk and Fly Navigation Mappings dialog box is never displayed.

**Display Position Locator Window** Specifies whether the Position Locator window opens when you enter walk mode.

**Current Drawing Settings** Specifies walk and fly mode settings specific to the current drawing.

**Walk/Fly Step Size** Sets the size of each step in drawing units (the *STEPSIZE* system variable).

**Steps Per Second** Specifies how many steps occur per second (the *STEPSPERSEC* system variable).

# WBLOCK

## Quick Reference

Writes objects or a block to a new drawing file

**wblock**

The Write Block dialog box (page 1547) is displayed.

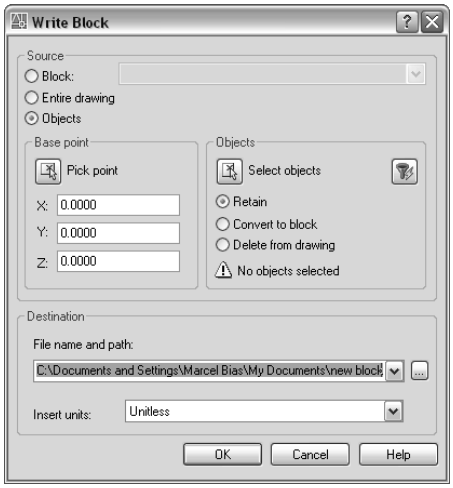
Entering **-wblock** at the Command prompt displays a standard file selection dialog box in which to specify a name for the new drawing file, followed by command prompts (page 1549). If *FILEDIA* is set to 0, the standard file selection dialog box is suppressed.

# Write Block Dialog Box

## Quick Reference

**wblock**

Saves objects or converts a block to a file.




The Write Block dialog box displays different default settings depending on whether nothing is selected, a single block is selected, or objects other than blocks are selected.

### Source

Specifies blocks and objects, saves them as a file, and specifies insertion points.

**Block** Specifies an existing block to save as a file. Select a name from the list.

**Entire Drawing** Selects current drawing as a block.

**Objects** Specifies a base point for the block. The default value is 0,0,0.

### Base Point

Specifies a base point for the block. The default value is 0,0,0.

**Pick Point** Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

**X** Specifies the *X* coordinate value for the base point.

**Y** Specifies the *Y* coordinate value for the base point.

**Z** Specifies the *Z* coordinate value for the base point.

### Objects

Sets the effect of block creation on objects used to create a block.

**Retain** Retains the selected objects in the current drawing after saving them as a file.

**Convert to Block** Converts the selected object or objects to a block in the current drawing after saving them as a file. The block is assigned the name in File Name.

**Delete from Drawing** Deletes the selected objects from the current drawing after saving them as a file.

**Select Objects Button** Temporarily closes the dialog box so that you can select one or more objects to save to the file.

**Quick Select Button** Opens the Quick Select dialog box (page 1152), which you can use to filter your selection set.

**Objects Selected** Indicates the number of objects selected.

### Destination

Specifies the new name and location of the file and the units of measurement to be used when the block is inserted.

**File Name and Path** Specifies a file name and path where the block or objects will be saved.

**[...]** Displays a standard file selection dialog box (page 931).

**Insert Units** Specifies the unit value to be used for automatic scaling when the new file is dragged from DesignCenter™ or inserted as a block in a drawing that uses different units. Select Unitless if you do not want to automatically scale the drawing when you insert it. See *INSUNITS*.

# -WBLOCK

## Quick Reference

If *FILEDIA* is set to 1, entering **-wblock** at the command prompt displays a standard file selection dialog box in which to specify a name for the new drawing file. If FILEDIA is set to 0, entering **-wblock** at the command prompt

displays a command prompt. The new drawing is saved in the file format that is specified in Save As on the Open and Save tab in the Options dialog box.

Enter name of output file:

Once you specify a filename—either through the dialog box or through the command prompt—the following prompt is displayed.

Enter name of existing block or
[= (block=output file)/* (whole drawing)] <define new drawing>: *Enter the name of an existing block, enter=, enter*, or press*ENTER

Entering the name of an existing block writes that block to a file. You cannot enter the name of an external reference (xref) or one of its dependent blocks.

Entering an equal sign (=) specifies that the existing block and the output file have the same name. If no block of that name exists in the drawing, the prompt is displayed again.

Entering an asterisk (*) writes the entire drawing to the new output file, except for unreferenced symbols. Model space objects are written to model space, and paper space objects are written to paper space.

If you press ENTER, you are prompted for an insertion base point for the block file and then prompts you to select the objects to write to the block file.

Specify insertion base point: *Specify a point (1)*
Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*


After the file is created, the selected objects are deleted from the drawing. You can use *OOPS* to restore the objects.

In the new drawing, the world coordinate system (WCS) is set parallel to the user coordinate system (UCS).

# WEBLIGHT

## Quick Reference

Creates a web light

**weblight**

Specify source location <0,0,0>: *Enter coordinate values or use the pointing device*
Specify target location <1,1,1>: *Enter coordinate values or use the pointing device*
Enter an option to change [Name (page 1551)/Intensity factor (page 1551)/Status (page 1551)/Photometry (page 1551)/weB (page 1552)/shadoW (page 1553)/filterColor (page 1553)/eXit (page 1554)] <eXit>:

**NOTE** The LIGHTINGUNITS system variable must be set to a value other than 0 to create and use weblights.

### Name

Specifies the name of the light. You can use uppercase and lowercase letters, numbers, spaces, hyphens (-), and underscores (_) in the name. The maximum length is 256 characters.

Enter light name:

### Intensity Factor

Sets the intensity or brightness of the light. The range is 0.00 to the maximum value that is supported by your system.

Enter intensity (0.00 - max float) <1.0000>:

### Status

Turns the light on and off. If lighting is not enabled in the drawing, this setting has no effect.

Enter status [oN/oFf]:

### Photometry

Photometry is available when the LIGHTINGUNITS system variable is set to 1 or 2. Photometry is the measurement of the luminous intensities of visible light sources.

In photometry, luminous intensity is a measure of the perceived power emitted by a light source in a particular direction. Luminous flux is the perceived power in per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions. Luminance is the total luminous flux incident on a surface, per unit area.

Enter a photometric option to change [Intensity/Color/eXit] <I>:
**Intensity** Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0000>:

Enter an intensity value in candelas, the perceived power in a luminous flux value, or illuminance value for the total luminous flux incident on a surface.

- Candela (symbol: cd) is the SI unit of luminous intensity (perceived power emitted by a light source in a particular direction). Cd/Sr
- Lux (symbol: lx) is the SI unit of illuminance. Lm/m^2
- Foot-candle (symbol: fc) is the American unit of illuminance. Lm/ft^2

Enter **f** to specify the perceived power in a luminous flux value.

Enter Flux (Lm) <18849.5556>:

If you enter **i**, you can specify the intensity of the light based on an illuminance value.

Enter Illuminance ("Lx"|"Fc") or enter an option [Distance] <1500.0000>:

The illuminance value can be specified in either lux or foot-candles. Enter **d** to specify a distance to use to calculate illuminance.

Enter Distance <1.0000>:

**Color** Enter color name or enter an option [?/Kelvin] <D65White>:

Specify the color of the light based on a color name or a Kelvin temperature. Enter ? to display a list of color names.

Enter color name(s) to list <*> :

Enter a text string using wild card characters to display a partial listing of color names, or an asterisk (*) to display all the possible choices.

If you enter **k**, you can specify the color of the light based on a Kelvin temperature value.

Enter Kelvin temperature <3600.0000>:

**Exit** Exits the command.

### Web

Specifies the intensity for a light at points on a spherical grid.

Enter a Web option to change [File/X/Y/Z/Exit] <Exit>:
**File** Enter Web file <>:

Specifies which web file to use to define the properties of the web. Web files have the file extension *.ies*.

**X** Enter Web X rotation <0.0000>:

Specifies the X rotation for the web.

**Y** Enter Web Y rotation <0.0000>:

Specifies the Y rotation for the web.

**Z** Enter Web Z rotation <0.0000>:

Specifies the Z rotation for the web.

**Exit** Exits the command option.

### Shadow

Makes the light cast shadows.

Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>:
**Off** Turns off the display and calculation of shadows for the light. Turning shadows off increases performance.

**Sharp** Displays shadows with sharp edges. Use this option to increase performance.

**Soft Mapped** Displays realistic shadows with soft edges.

Enter map size [64/128/256/512/1024/2048/4096] <256>:

Specifies the amount of memory to use to calculate the shadow map.

Enter softness (1-10) <1>:

Specifies the softness to use to calculate the shadow map.

**Soft Sampled** Displays realistic shadows with softer shadows (penumbra) based on extended light sources.

Enter an option to change [Shape/sAmples/Visible/eXit]<eXit> :

Specify the shape of the shadow by entering **s** and then the dimensions of the shape. (For example, the radius of the sphere or the length and width of a rectangle.)

Enter shape [Linear, Disk, Rect, Sphere, Cyl] <Rect>:

Specify the sample size by entering **a**.

Enter Shadow Sample <16.0000>:

Specify the visibility of the shape by for the shadow by entering **v.**

Enter Shape Visibility [Yes/No]<No>:

### Filter Color

Controls the color of the light.

Enter true color (R,G,B) or enter an option [Index color/Hsl/colorBook]:
**True Color** Specifies a True Color. Enter in the format R,G,B (red, green, blue).

**Index** Specifies an ACI (AutoCAD Color Index) color.

Enter color name or number (1-255):

**HSL** Specifies an HSL (hue, saturation, luminance) color.

Enter HSL color (H,S,L):

**Color Book** Specifies a color from a color book.

Enter Color Book name:

#### Exit

Exits the command.

# WEDGE

## Quick Reference

Creates a five-sided 3D solid with a sloped face tapering along the X axis

Modeling

Draw ➤ Modeling ➤ WedgeAt the Command prompt, enter wedge.

**wedge**

3D Make panel, Wedge

Specify first corner or [Center (page 1554)]: *Specify a point or enter* **c** *for center*

Specify other corner or [Cube (page 1555)/Length (page 1556)]: *Specify the other corner of the wedge or enteran option*

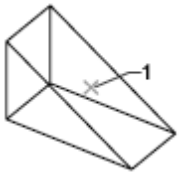
If the other corner of the wedge is specified with a *Z* value that differs from the first corner, then no height prompt is displayed.

Specify height or [2Point (page 1556)] <default>: *Specify the height or enter* **2P** *for the 2 Point option*

Entering a positive value draws the height along the positive *Z* axis of the current UCS. Entering a negative value draws the height along the negative *Z* axis.

#### Center

Creates the wedge by using a specified center point.




Specify center: *Specify a point (1)*
Specify other corner or [Cube/Length]: *Specify a point or enter an option*
Specify height or [2Point] <default>: *Specify the height or enter* **2P** *for the 2 Point option*
**Cube** Creates a wedge with sides of equal length.

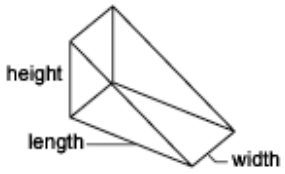
Specify length: *Enter a value, or pick a point to specify the length and the rotation of the wedge in the XY plane*

**Length** Creates a wedge with length, width, and height values you specify. The length corresponds to the *X* axis, the width to the *Y* axis, and the height to the *Z* axis. If you pick a point to specify the length, you also specify the rotation in the *XY* plane.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the wedge in the XY plane*

Specify width: *Specify a distance*

Specify height: *Specify a distance*




### Cube

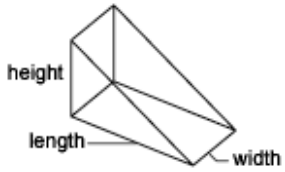
Creates a wedge with sides of equal length.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the wedge in the XY plane*

**Length**

Creates a wedge with length, width, and height values you specify. The length corresponds to the *X* axis, the width to the *Y* axis, and the height to the *Z* axis.

Specify length: *Enter a value, or pick a point to specify the length and the rotation of the wedge in the XY plane*
Specify width: *Specify a distance*
Specify height: *Specify a distance*




**2Point**

Specifies that the height of the wedge is the distance between the two specified points.

Specify first point: *Specify a point*
Specify second point: *Specify a point*

# WHOHAS

## Quick Reference

Displays ownership information for opened drawing files

**whohas**

You can use WHOHAS to track which users have certain drawing files open.

WHOHAS displays the Select Drawing to Query dialog box (a standard file selection dialog box (page 931)). After you select a file, the ownership

information is displayed at the command prompt. The information includes the current user's computer name, login ID, and full name (if available) and the date and time the drawing file was opened.

**NOTE** Similar information is displayed automatically when you try to open a drawing file that another user has already opened.

The information displayed by WHOHAS is stored in a temporary DWL (drawing lock) file. A DWL file is automatically created when a file is opened and then deleted when the file is closed.

# WIPEOUT

## Quick Reference

Covers existing objects with a blank area

Draw ➤ WipeoutAt the Command prompt, enter wipeout.
**wipeout**

Creates a polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame. You can turn on the wipeout frame for editing and turn it off for plotting.

Specify first point or [Frames/Polyline]<Polyline>: *Specify a point or an option*
**First Point** Determines the polygonal boundary of the wipeout object from a series of points.

Next point: *Specify the next point or press* ENTER *to exit*

**Frames** Determines whether the edges of all wipeout objects are displayed or hidden.

Enter mode [ON/OFF]:<*varies*> *Enter* **on** *or* **off**

Enter **on** to display all wipeout frames. Enter **off** to suppress the display of all wipeout frames.

**Polyline** Determines the polygonal boundary of the wipeout objects from a selected polyline.

Select a closed polyline: *Use an object selection method to select a closed polyline*

Erase polyline? [Yes/No]<N>: *Enter* **y** *or* **n**

Enter **y** to erase the polyline that was used to create the wipeout object. Enter **n** to retain the polyline.

# WMFIN

## Quick Reference

Imports a Windows metafile

Insert ➤ Windows MetafileAt the Command prompt, enter wmfin.
**wmfin**

The Import WMF dialog box (a standard file selection dialog box (page 931)) is displayed. Enter a file name. The *.wmf* file extension is added automatically.

In the Import WMF dialog box, if you click Tools ➤ Options, the WMF In Options dialog box (page 1562) is displayed. You can also open this dialog box directly by using *WMFOPTS*. After opening the selected WMF file, the following prompt is displayed:

Specify insertion point (page 1558) or [Scale (page 1559)/X (page 1559)/Y (page 1559)/Z (page 1560)/Rotate (page 1560)/PScale (page 1560)/PX (page 1561)/PY (page 1561)/PZ (page 1561)/PRotate (page 1562)]: *Specify a point or enter an option*

### Insertion Point

Places a copy of the metafile with its base point at the specified insertion point.

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER
**X Scale Factor** Sets the *X* and *Y* scale factors.

Enter Y scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Specify rotation angle <0>:

All *X* and *Y* dimensions of the metafile are multiplied by the *X* and *Y* scale factors supplied. The metafile is rotated by the angle specified, using the insertion point as the center of rotation.

**Corner** Defines the X and Y scales at the same time, using the insertion point and another point as the corners of a box. The X and Y dimensions of the box become the X and Y scale factors. The insertion point is the first corner.

Specify opposite corner: *Specify a point*

Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

**XYZ** Scales the metafile in all three dimensions.

Specify X scale factor or [Corner] <1>: *Specify a nonzero value, enter* **c**, *or press* ENTER

If you enter **c**, you specify a corner point. The specified point and the insertion point determine the *X* and *Y* scale factors for the WMF file.

Specify Y scale factor or <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter Z scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

### Scale

Sets the scale factor for the *X*, *Y*, and *Z* axes. The scale for the *Z* axis is the absolute value of the specified scale factor.

Specify scale factor for XYZ axes: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

### X

Sets the *X* scale factor.

Specify X scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

### Y

Sets the *Y* scale factor.

Specify Y scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

### Z

Sets the *Z* scale factor.

Specify Z scale factor: *Enter a nonzero value*
Specify insertion point:
Specify rotation angle <0>:

The rotation angle sets the angle of the WMF file.

### Rotate

Sets the angle of insertion for the WMF file.

Specify rotation angle <0>:
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER
**X Scale Factor** Sets the *X* and *Y* scale factors.

Enter Y scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

**Corner** Defines the *X* and *Y* scales at the same time, using the insertion point and another point as the corners of a box. The *X* and *Y* dimensions of the box become the X and Y scale factors. The insertion point is the first corner.

Specify opposite corner: *Specify a point*

**XYZ** Scales the metafile in all three dimensions.

Specify X scale factor or [Corner] <1>: *Specify a nonzero value, enter* **c**, *or press* ENTER

If you enter **c**, you specify a corner point. The specified point and the insertion point determine the *X* and *Y* scale factors for the block.

Specify Y scale factor or <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter Z scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

### PScale

Sets the temporary scale factor for the *X*, *Y*, and *Z* axes to control the display of the WMF file as it is dragged into position.

Specify preview scale factor for XYZ axes: *Enter a nonzero value*
Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match the descriptions of the corresponding options under Rotate.

### PX

Sets the temporary scale factor for the *X* axis to control the display of the WMF file as it is dragged into position.

Specify preview X scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match the descriptions of the corresponding options under Rotate.

### PY

Sets the temporary scale factor for the *Y* axis to control the display of the WMF file as it is dragged into position.

Specify preview Y scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match the descriptions of the corresponding options under Rotate.

### PZ

Sets the temporary scale factor for the *Z* axis to control the display of the WMF file as it is dragged into position.

Specify preview Z scale factor: *Enter a nonzero value*
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match the descriptions of the corresponding options under Rotate.

**PRotate**

Sets the temporary rotation angle of the WMF file as it is dragged into position.

Specify preview rotation angle:
Specify insertion point:
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, specify a point, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match the descriptions of the corresponding options under Rotate.

# WMFOPTS

## Quick Reference

Sets options for WMFIN

**wmfopts**

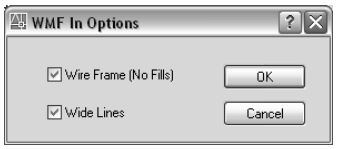
The WMF In Options dialog box (page 1562) is displayed.

# WMF In Options Dialog Box

## Quick Reference

**wmfopts**

Controls whether metafiles maintain relative line widths and whether they are imported as wireframes or solid objects.




**Wire Frame (No Fills)** Imports objects as wireframes. If you clear this option, objects are imported as filled objects.

**Wide Lines** Maintains the relative line width of lines and borders. If you clear this option, lines are imported with zero width.

# WMFOUT

## Quick Reference

Saves objects to a Windows metafile

**wmfout**

The Create WMF standard file selection dialog box (page 931) is displayed. Enter a file name. A *.wmf* file extension is added.

Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*

The selected objects are saved to a file in Windows metafile format.

# WORKSPACE

## Quick Reference

Creates, modifies, and saves workspaces and makes a workspace current

**workspace**
Enter Workspace option [setCurrent/SAveas/Edit/Rename/Delete/SEttings/?] <setCurrent>:
**Set Current** Sets a current workspace.

Enter name of workspace to make current [?] <*current*>: *Enter a name or* **?** *to list available workspaces.*

**Save As** Saves a current interface configuration as a workspace.

Save workspace as <*current*>:

**Edit** Opens the Customize User Interface dialog box, Customize tab (page 277), where you can make modifications to a workspace.

**Rename** Renames a workspace.

Enter workspace to rename [?] <*current*>

Enter new workspace name <*current*>:

Workspace <*name*> renamed to <*new_name*>.

**Delete** Deletes a workspace.

Enter workspace to delete *[?]*: *Enter a name or* **?** *to list available workspaces.*

Do you really want to delete the workspace <*name*> [Yes/No]? <N>

Workspace <*name*> deleted.

**Settings** Opens the Workspace Settings dialog box (page 1564), which controls the display, menu order, and Save settings of a workspace.

**?—List Workspaces** Lists all workspaces defined in the main and enterprise CUI files.

# WSSETTINGS

## Quick Reference

Sets options for workspaces

**wssettings**

The Workspace Settings dialog box (page 1564) is displayed.

# Workspace Settings Dialog Box

## Quick Reference

**wssettings**

Controls the display, menu order, and Save settings of a workspace.

**My Workspace =** Displays a list of workspaces from which you can choose a workspace to assign to the My Workspace toolbar button.

**Menu Display and Order** Controls which workspace name you want to display in the Workspaces toolbar and menu, the order of those workspace names, and whether a separator line is added between each workspace name. The workspaces displayed here and in the Workspaces toolbar and menu include the current workspace (displayed with a check mark in the toolbar and menu) and the workspace you've defined in the My Workspace= option, regardless of the display settings.

**Move Up** Moves workspace names up in the display order.

**Move Down** Moves workspace names down in the display order.

**Add Separator** Adds a separator between workspace names.

**Do Not Save Changes to Workspace** Does not save changes you've made to a workspace when you switch to another workspace

**Automatically Save Workspace Changes** Saves changes you've made to a workspace when you switch to another workspace.

# WSSAVE

## Quick Reference

Saves a workspace

**wssave**

If you enter **-wssave** at the command prompt, WSSAVE displays command prompts.

The Save Workspace dialog box (page 1565) is displayed.

# Save Workspace Dialog Box

## Quick Reference

**wssave**

Saves the current workspace scheme and settings of a workspace.

**Name** Displays a text box where you can specify a name for a newly saved workspace and view a list of existing workspaces. These workspaces can be overwritten if the CUI file in which they are included is writable. Read-only files, such as CUI files that are shared across a network (called *enterprise* CUI files), are displayed with a "Read-Only" file name extension.

# X Commands

# 24

**In this chapter**

- XATTACH
- XBIND
- XCLIP
- XEDGES
- XLINE
- XOPEN
- XPLODE
- XREF

# XATTACH

### Quick Reference

Attaches an external reference to the current drawing

Reference 
Insert ➤ External ReferenceDoes not exist in the menus.
**xattach**

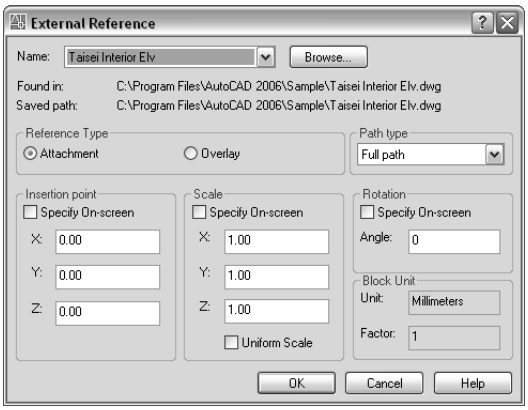
The Select Reference File dialog box (a standard file selection dialog box (page 931)) is displayed. The External Reference dialog box (page 1568) is displayed after a file is selected.

## External Reference Dialog Box

### Quick Reference

Reference
Insert ➤ External ReferenceDoes not exist in the menus.
**xattach**

Attaches a drawing as an external reference (xref). If you attach a drawing that itself contains an attached xref, the attached xref appears in the current drawing. Like blocks, attached xrefs can be nested. If another person is currently editing the xref, the drawing attached is based on the most recently saved version.




### Name

Displays the xref name in the list after an xref is attached. When an attached xref is selected in the list, its path is displayed in Path.

### Browse

Choose Browse to display the Select a Reference dialog box (a standard file selection dialog box (page 931)), in which you can select new xrefs for the current drawing.

### Found In

Displays the path where the xref was found.

If no path was saved for the xref or if the xref is no longer located at the specified path, the program searches for the xref in the following order:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the *PROJECTNAME* system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start-in folder specified in the Microsoft® Windows® application shortcut

### Saved Path

Displays the saved path, if any, that is used to locate the xref. This path can be a full (absolute) path, a relative (partially specified) path, or no path.

### Reference Type

Specifies whether the xref is an attachment or an overlay. Unlike an xref that is an attachment, an overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

See "Attach Drawing References (Xrefs)" and "Nest and Overlay Referenced Drawings" in the *User's Guide*.

### Path Type

Specifies whether the saved path to the xref is the full path, a relative path, or no path. You must save the current drawing before you can set the path type to Relative Path. For a nested xref, a relative path always references the location of its immediate host and not necessarily the currently open drawing.

The Relative Path option is not available if the referenced drawing is located on a different local disk drive or on a network server.

### Insertion Point

Specifies the insertion point of the selected xref.

**Specify On-Screen** Displays command prompts and makes the X, Y, and Z options unavailable. The descriptions of the options displayed at the command prompt match the descriptions of the corresponding options under Insertion Point in *INSERT*.

**X** Specifies the *X* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the *BASE* command in the referenced file.

**Y** Specifies the *Y* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the BASE system variable in the referenced file.

**Z** Specifies the *Z* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the BASE system variable in the referenced file.

### Scale

Specifies the scale factors for the selected xref.

**Specify On-Screen** Displays command prompts and makes the X, Y, and Z Scale Factor options unavailable. The descriptions of the options displayed at the command prompt match the descriptions of the corresponding options under Insertion Point in *INSERT*.

**X Scale Factor** Specifies the *X* scale factor for the xref instance.

**Y Scale Factor** Specifies the *Y* scale factor for the xref instance.

**Z Scale Factor** Specifies the *Z* scale factor for the xref instance.

**Uniform Scale** Ensures that the *Y* and *Z* scale factors are equal to the *X* scale factor.

#### Rotation

Specifies the rotation angle for the xref instance.

**Specify On-Screen** Displays command prompts and makes the Angle option unavailable. You are prompted for a rotation angle, as described in *INSERT*.

**Angle** Specifies the rotation angle at which an xref instance is inserted into the current drawing.

#### Block Unit

Displays information about the block units.

**Unit** Displays the specified *INSUNITS* value for the inserted block.

**Factor** Displays the unit scale factor, which is calculated based on the *INSUNITS* value of the block and the drawing units.

# XBIND

## Quick Reference

Binds one or more definitions of named objects in an xref to the current drawing

Reference 
Modify ➤ Object ➤ External Reference ➤ BindAt the Command prompt, enter xbind.

**xbind**

The Xbind dialog box (page 1572) is displayed.

If you enter **-xbind** at the command prompt, options are displayed at the command prompt (page 1573).

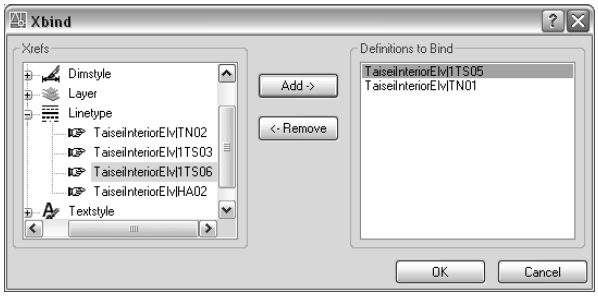
**NOTE** The Bind option of XREF binds the xref file. Use XBIND for individual dependent definitions.

# Xbind Dialog Box

## Quick Reference

Reference
Modify ➤ Object ➤ External Reference ➤ BindAt the Command prompt, enter xbind.
**xbind**

Adds xref-dependent named objects (such as blocks, dimension styles, layers, linetypes, and text styles) to your drawing.




**Xrefs** Lists the xrefs currently attached to the drawing. Selecting an xref (double-clicking) displays the named object definitions in the attached xref.

**Definitions to Bind** Lists the xref-dependent named object definitions to bind to the host drawing.

**Add** Moves the named object definitions selected in the Xrefs list into the Definitions to Bind list.

**Remove** Moves the xref-dependent named object definition selected in the Definitions to Bind list back to its xref-dependent definition table.

# -XBIND

## Quick Reference

If you enter **-xbind** at the command prompt, the following XBIND command prompts are displayed.

Enter symbol type to bind [Block/Dimstyle/LAyer/LType/Style]: *Enter an option*

Depending on the option, you are prompted for a xref-dependent named object (symbol) such as a block, dimension style, layer, linetype, or text style.

Enter dependent *Symbol* name(s): *Enter a name list or * to bind all xref-dependent named objects (symbols) from that definition table*

The name you specify must be the full name, including the vertical bar character ( | ).

The xref-dependent named objects you specify are added to your drawing. You can manipulate them as you would any other named object. The vertical bar character ( | ) from each xref-dependent named object is replaced with three new characters: a number (usually 0) between two dollar signs ($).

If you specify a layer whose associated linetype is not CONTINUOUS, XBIND also binds the referenced linetype. If you apply XBIND to a block, any block, dimension style, layer, linetype, or text style that's referenced by objects in the block is also bound. If the block contains an xref, XBIND binds that xref and all its dependent named objects.

# XCLIP

## Quick Reference

Defines an xref or block clipping boundary and sets the front and back clipping planes

Reference 
Select an xref, right-click in the drawing area, and choose Xref Clip.
**xclip**
Select objects: *Use an object selection method and press ENTER* when you finish selecting objects

Enter clipping option [ON (page 1574)/OFF (page 1574)/Clipdepth (page 1574)/Delete (page 1574)/generate Polyline (page 1574)/New boundary (page 1575)] <New>: *Select an option or press* ENTER

### On

Displays the clipped portion of the xref or block in the current drawing.

### Off

Displays all of the geometry of the xref or block in the current drawing, ignoring the clipping boundary.

### Clipdepth

Sets the front and back clipping planes on an xref or block. Objects outside the volume defined by the boundary and the specified depth are not displayed. Regardless of the current UCS, the clip depth is applied parallel to the clipping boundary.

Specify front clip point or [Distance/Remove]:
**Front Clip Point** Creates a clipping plane passing through and perpendicular to the clipping boundary.

Specify back clip point or [Distance/Remove]:

**Distance** Creates a clipping plane the specified distance from and parallel to the clipping boundary.

Specify distance from boundary:

The previous prompt is displayed.

**Remove** Removes both the front and back clipping planes.

### Delete

Removes a clipping boundary for the selected xref or block. To temporarily turn off a clipping boundary, use the Off option. Delete erases the clipping boundary and the clipdepth. The *ERASE* command cannot be used to delete clipping boundaries.

### Generate Polyline

Automatically draws a polyline coincident with the clipping boundary. The polyline assumes the current layer, linetype, lineweight, and color settings. Use this option when you want to modify the current clipping boundary using

*PEDIT* and then redefine the clipping boundary with the new polyline. To see the entire xref while redefining the boundary, use the Off option.

### New Boundary

Defines a rectangular or polygonal clipping boundary or generates a polygonal clipping boundary from a polyline. If a boundary already exists, the following prompt is displayed:

Delete old boundary? [Yes/No] <Yes>:

The command continues only if all previous boundaries are deleted.

*Current* mode: Objects *inside/outside* boundary will be hidden.
Specify clipping boundary:
[Select polyline/Polygonal/Rectangular/Invert clip] <Rectangular>: *Specify a clipping boundary option or press* ENTER
**Select Polyline** Defines the boundary using the selected polyline. The polyline can be open but must consist of straight line segments and cannot intersect itself. The boundary created using this method is parallel to the user coordinate system (UCS) plane on which the polyline lies.

**Polygonal** Defines a polygonal boundary by using the points you specify for the vertices of a polygon. The clipping boundary is applied in the current UCS regardless of the current view.

**Rectangular** Defines a rectangular boundary by using the points you specify for opposite corners. The clipping boundary is applied in the current UCS and is independent of the current view.

**Invert Clip** Inverts the mode of the clipping boundary: either the objects outside the boundary (default) or inside the boundary are hidden.

---

**NOTE** Use the 2D Wireframe visual style for viewing inverted clipped xrefs and blocks. Other visual styles do not display inverted clipped areas.

---

# XEDGES

## Quick Reference

Creates wireframe geometry by extracting edges from a 3D solid or surface

Modify ➤ 3D Operations ➤ Extract EdgesAt the Command prompt, enter xedges.

**xedges**

With the XEDGES command, you can create wireframe geometry by extracting all the edges from the following objects:

- Solids
- Regions
- Surfaces

You can also select individual edges and faces to extract. Hold down the CTRL key to select edges and faces.

Select objects: *Select objects from which to extract wireframe geometry and then press* ENTER

# XLINE

## Quick Reference

Creates an infinite line

Draw 

Draw ➤ Construction LineAt the Command prompt, enter xline.

**xline**

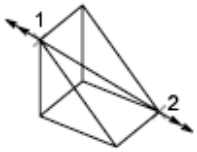
Specify a point (page 1576) or [Hor (page 1577)/Ver (page 1577)/Ang (page 1577)/Bisect (page 1578)/Offset (page 1578)]: *Specify a point or enter an option*

**Point**

Specifies the location of the infinite line using two points through which it passes.

Specify through point: *Specify the point (2) through which you want the xline to pass, or press* ENTER *to end the command*

The xline is created through the specified point.




### Hor

Creates a horizontal xline passing through a specified point.

Specify through point: *Specify the point (1) through which you want the xline to pass, or press* ENTER *to end the command*

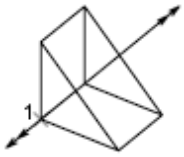
The xline is created parallel to the *X* axis.


### Ver

Creates a vertical xline passing through a specified point.

Specify through point: *Specify the point (1) through which you want the xline to pass, or press* ENTER *to end the command*

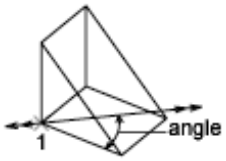
The xline is created parallel to the *Y* axis.




### Ang

Creates an xline at a specified angle.

Enter angle of xline (0) or [Reference]: *Specify an angle or enter* **r**




**Angle of Xline** Specifies the angle at which to place the line.

Specify through point: *Specify the point through which you want the xline to pass*

An xline is created through the specified point, using the specified angle.

**Reference** Specifies the angle from a selected reference line. The angle is measured counterclockwise from the reference line.

Select a line object: *Select a line, polyline, ray, or xline*

Enter angle of xline <0>:

Specify through point: *Specify the point through which you want the xline to pass, or press* ENTER *to end the command*

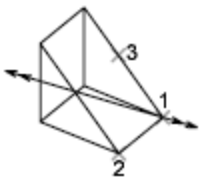
An xline is created through the specified point, using the specified angle.

### Bisect

Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line.

Specify angle vertex point: *Specify a point (1)*
Specify angle start point: *Specify a point (2)*
Specify angle end point: *Specify a point (3) or press* ENTER *to end the command*

The xline lies in the plane determined by the three points.




### Offset

Creates an xline parallel to another object.

Specify offset distance or [Through] <*current*>: *Specify an offset distance, enter* **t**, *or press* ENTER

**Offset Distance** Specifies the distance the xline is offset from the selected object.

Select a line object: *Select a line, polyline, ray, or xline, or press* ENTER *to end the command*

Specify side to offset? *Specify a point and press* ENTER *to exit the command*

**Through** Creates an xline offset from a line and passing through a specified point.

Select a line object: *Select a line, polyline, ray, or xline, or press* ENTER *to end the command*

Specify through point: *Specify the point through which you want the xline to pass and press* ENTER *to exit the command*

# XOPEN

## Quick Reference

Opens a selected drawing reference (xref) in a new window

**xopen**
Select Xref: *Select an object in the drawing reference*

After an object is selected, the drawing reference to which it belongs opens in a separate window. If the drawing reference contains nested xrefs, the deepest-level drawing reference of the selected object is opened.

# XPLODE

## Quick Reference

Breaks a compound object into its component objects

**xplode**
Select objects to XPlode.
Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*

The program reports how many objects were selected and, of those, how many objects cannot be exploded. If you select more than one valid object, the following prompt is displayed.

Enter an option [Individually (page 1580)/Globally (page 1581)] <Globally>: *Enter* **i**, *enter* **g**, *or press* ENTER


### Individually

Applies changes to the selected objects one at a time. The following prompt is displayed for each object.

Enter an option [All/Color/LAyer/LType/LWeight/Inherit from parent block/Explode] <Explode>: *Enter an option or press* ENTER
**All** Sets the color, linetype, lineweight, and layer of the component objects after you explode them. The prompts associated with the Color, Linetype, Lineweight, and Layer options are displayed.

**Color** Sets the color of the objects after you explode them.

New color [Truecolor/COlorbook]<BYLAYER>: *Enter a color name or number from* **1** *through* **255**, *enter* **t**, *enter* **co**, *enter* **bylayer** *or* **byblock**, *or press* ENTER

Entering **bylayer** causes the component objects to inherit the color of the exploded object's layer. Entering **byblock** causes the component objects to inherit the color of the exploded object.

Enter **t** for a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from* **0** *to* **255** *separated by commas to specify a true color*

Enter **co** for a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE®**

If you enter a color book name, you are prompted to enter the color name in the color book.

Enter color name: *Enter the name of a color included in the selected color book, such as* **PANTONE573**

**Layer** Sets the layer of the component objects after you explode them. The default option is to inherit the current layer rather than the layer of the exploded object.

Enter new layer name for exploded objects <*current*>: *Enter an existing layer name or press* ENTER

**LType** Sets the linetype of the component objects after you explode them.

Enter new linetype name for exploded objects <BYLAYER>: *Enter a linetype name, or press* ENTER

You can enter the name of any linetype that is loaded in the drawing. Entering **bylayer** causes the component objects to inherit the linetype of the exploded object's layer. Entering **byblock** causes the component objects to inherit the linetype of the exploded object.

**LWeight** Sets the lineweight of the component objects after you explode them.

Enter new lineweight for exploded objects <BYLAYER>: *Enter a numeric distance or two points, or press* ENTER

**Inherit from Parent Block** Sets the color, linetype, lineweight, and layer of the component objects to that of the exploded object if the component objects' color, linetype, and lineweight are BYBLOCK and the objects are drawn on layer 0.

**Explode** Breaks a compound object into its component objects exactly as the *EXPLODE* command does.

#### Globally

Applies changes to all the selected objects.

Enter an option [All/Color/LAyer/LType/Inherit from parent block/Explode] <Explode>: *Enter an option or press* ENTER

The descriptions of the All, Color, Layer, LType, Inherit from Parent Block, and Explode options match the descriptions of the corresponding options under Individually.

# XREF

## Quick Reference

Starts the EXTERNALREFERENCES command

**xref**

The External References palette (page 561) is displayed.

If you enter **-xref** at the command prompt, options are displayed at the command prompt (page 1583).

# Bind Xrefs Dialog Box

## Quick Reference

Reference
Insert ➤ External ReferencesAt the Command prompt, enter externalreferences.
Select a DWG reference (xref), right-click in the drawing area, and choose External References.
externalreferences

Converts DWG references (xrefs) to standard local block definitions. If you bind an xref into the current drawing, the xref and all its dependent named objects become a part of the current drawing. Use *XBIND* to add individual xref-dependent named objects, such as blocks, text styles, dimension styles, layers, and linetypes, to the local definition table. The two methods of binding xrefs to the current drawing are Bind and Insert. Bind alters the definition table names of an xref when it is inserted. Insert does not alter the definition table names of an xref when it is inserted. To bind a nested xref, you must also select the parent xref.


**Bind** Binds the selected DWG reference to the current drawing. Xref-dependent named objects are changed from *blockname|definitionname* to *blockname$n$definitionname* syntax. In this manner, unique named objects are created for all xref-dependent definition tables bound to the current drawing.

For example, if you have an xref named FLOOR1 containing a layer named WALL, after binding the xref, the xref-dependent layer FLOOR1|WALL becomes a locally defined layer named FLOOR1$0$WALL. The number in $*n*$ is automatically incremented if a local named object with the same name already exists. In this example, if FLOOR1$0$WALL already existed in the drawing, the xref-dependent layer FLOOR1|WALL would be renamed FLOOR1$1$WALL.

**Insert** Binds the DWG reference to the current drawing in a way similar to detaching and inserting the reference drawing. Rather than being renamed using *blockname$n$definitionname* syntax, xref-dependent named objects are stripped of the xref name. As with inserting drawings, no name-incrementing occurs if a local named object shares the same name as a bound xref-dependent

named object. The bound xref-dependent named object assumes the properties of the locally defined named object.

For example, if you have an xref named FLOOR1 containing a layer named WALL, after binding with the Insert option, the xref-dependent layer FLOOR1|WALL becomes the locally defined layer WALL.

# -XREF

## Quick Reference

If you enter **-xref** at the command prompt, the following XREF command prompts are displayed.

Enter an option [? (page 1583)/Bind (page 1583)/Detach (page 1583)/Path (page 1584)/Unload (page 1584)/Reload (page 1584)/Overlay (page 1584)/Attach (page 1584)] <Attach>: *Enter an option or press* ENTER

**?—List Xrefs** Lists the DWG reference name, path, and type and the number of DWG references currently attached to your drawing. The following prompt is displayed:

Enter xref name(s) to list <*>: *Enter a name list or press* ENTER *to list all xrefs in the drawing*

**Bind** Converts a specified DWG reference into a block, making it a permanent part of the drawing.

Enter xref name(s) to bind: Enter a name or a list of names separated by commas

The xref-dependent named objects, such as layer names, of the former xref are added to your drawing. In each xref-dependent named object, the vertical bar (|) is replaced with three new characters: a number (usually 0) between two dollar signs ($). The number is incremented if the same name already exists in the current drawing.

**Detach** Detaches one or more DWG references from your drawing, erasing all instances of a specified xref and marking the xref definition for deletion from the definition table. Only the xrefs attached or overlaid directly to the current drawing can be detached; nested xrefs cannot be detached.

Enter xref name(s) to detach:

An xref referenced by another xref or block cannot be detached.

**Path** Displays and edits the path name associated with a particular DWG reference. This option is useful if you change the location of or rename the drawing file associated with the xref.

Edit xref name(s) to edit path:

The xref and its old path are listed, and you are prompted for the new path:

xref name: *"name"*

Old path: *path*

Enter new path: *Enter new path and drawing name*

If you enter an invalid path, or the drawing cannot be found at the location you enter, you are prompted for the new path name.

**Unload** Unloads the selected DWG references.

Enter xref name(s) to unload:

A marker is left in place of the xref so that it can be reloaded later.

**Reload** Reloads one or more DWG references. This option reloads and displays the most recently saved version of that drawing.

Enter xref name(s) to reload:

If the program encounters an error while reloading, it ends XREF and undoes the entire reloading sequence.

**Overlay** Displays the Enter Name of File to Overlay dialog box (a standard file selection dialog box). Select the file you want to attach to a drawing as an external reference (xref) overlay. If you reference a drawing that contains an overlaid xref, the overlaid xref does not appear in the current drawing.

Unlike blocks and attached xrefs, overlaid xrefs cannot be nested. If another person is currently editing the xref file, the program overlays the most recently saved version.

If the xref you specify is not already overlaid, a new xref is crated, using the name of the referenced file. You are then prompted for an insertion point, scale, and rotation angle, as described for the *INSERT* command.

If FILEDIA is set to 0, the following prompt is displayed:

Enter name of file to overlay:

You can enter a tilde (~) to display a dialog box.

**Attach** Displays the External Reference dialog box, if a DWG reference is selected, or the Select Reference File dialog box, if no DWG reference is selected. See *XATTACH*.

If you reference a drawing that contains an attached xref, the attached xref appears in the current drawing. Like blocks, attached xrefs can be nested. If

another person is currently editing the xref file, the most recently saved version is attached.

If the xref you specify is not already attached or overlaid, a new xref is created, using the name of the referenced file. You are prompted for an insertion point, scale, and rotation angle, as described for the *INSERT* command.

When reading objects from the xref into the new block, the program copies only those objects created in model space and ignores objects created in paper space. Thus, viewports and other objects in paper space do not become a part of the block definition.

In a master drawing, the 0 and DEFPOINTS layers and the CONTINUOUS linetype override entries with the same name that exist in the attaching or overlaying xref. Any objects on these layers remain unaffected. If the DEFPOINTS layer exists in the attaching or overlaying xref but not in the master drawing, the DEFPOINTS layer becomes a permanent part of the master drawing.

If FILEDIA is set to 0, the following prompt is displayed:

Enter name of file to attach:

You can enter a tilde (~) to display a dialog box.

# Z Commands

# 25

**In this chapter**

- ZOOM

# ZOOM

## Quick Reference

Increases or decreases the apparent size of objects in the current viewport

Standard 

View ➤ ZoomDoes not exist in the menus.

With no objects selected, right-click in the drawing area and choose Zoom to zoom in real time.

**zoom** (or '**zoom** for transparent use)

In a perspective view, ZOOM displays the *3DZOOM* prompts.

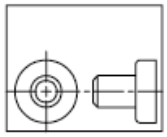
---

**NOTE** You cannot use ZOOM transparently during *VPOINT* or *DVIEW* or while ZOOM, *PAN*, or *VIEW* is in progress.

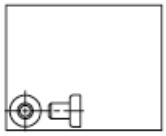
---

Specify corner of window, enter a scale factor (nX or nXP), or [All (page 1588)/Center (page 1588)/Dynamic (page 1589)/Extents (page 1589)/Previous (page 1589)/Scale (page 1590)/Window (page 1590)/Object (page 1591)] <real time (page 1591)>:

**All** Zooms to display the entire drawing in the current viewport. In a plan view, All zooms to the grid limits or current extents, whichever is greater. In a 3D view, ZOOM All is equivalent to ZOOM Extents. The display shows all objects even if the drawing extends outside the grid limits.



before ZOOM All



after ZOOM All

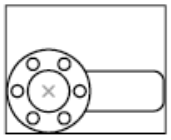
In the illustration, *LIMITS* is greater than the extents of the drawing.

Because it always regenerates the drawing, you cannot use ZOOM All transparently.

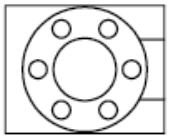
**Center** Zooms to display a window defined by a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the magnification.

Specify center point: *Specify a point (1)*

Enter magnification or height <*current*>: *Enter a value or press* ENTER

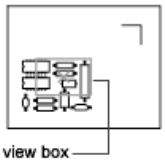


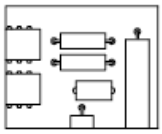
before ZOOM center



after ZOOM center,
magnification increased

**Dynamic** Zooms to display the generated portion of the drawing with a view box. The view box represents your viewport, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms to fill the viewport with the image inside the view box.



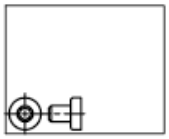



new view

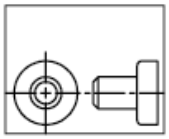
The panning view box is displayed first. Drag it to the location you want and click. The zooming view box is then displayed. Resize it and press ENTER to zoom, or click to return to the panning view box.

Press ENTER to fill the current viewport with the area currently enclosed by the view box.

**Extents** Zooms to display the drawing extents and results in the largest possible display of all the objects.

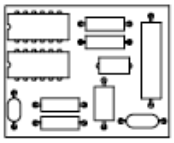


before ZOOM Extents

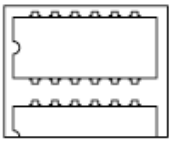


after ZOOM Extents

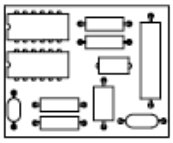
**Previous** Zooms to display the previous view. You can restore up to 10 previous views.



original view



current view



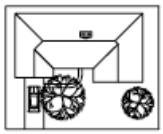
after ZOOM Previous

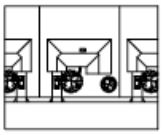
**NOTE** If you change the visual style, the view is changed. If you enter ZOOM Previous, it restores the previous view, which is shaded differently but not zoomed differently.

**Scale** Zooms the display at a specified scale factor.

Enter a scale factor (nX or nXP): *Specify a value*

Enter a value followed by **x** to specify the scale relative to the current view. For example, entering **.5x** causes each object to be displayed at half its current size on the screen.

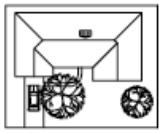


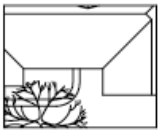


ZOOM .5x

Enter a value followed by **xp** to specify the scale relative to paper space units. For example, entering **.5xp** displays model space at half the scale of paper space units. You can create a layout with each viewport displaying objects at a different scale.

Enter a value to specify the scale relative to the limits of the drawing. (This option is rarely used.) For example, entering **2** displays objects at twice the size they would appear if you were zoomed to the limits of the drawing.



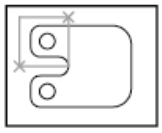


ZOOM 2

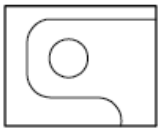
**Window** Zooms to display an area specified by two opposite corners of a rectangular window.

Specify first corner: *Specify a point (1)*

Specify opposite corner: *Specify a point (2)*



before ZOOM Window



after ZOOM Window

**Object** Zooms to display one or more selected objects as large as possible and in the center of the drawing area. You can select objects before or after you start the ZOOM command.

**Real Time** Using the pointing device, zooms interactively to a logical extent.

Press ESC or ENTER to exit, or right-click to display the shortcut menu

The cursor changes to a magnifying glass with plus (+) and minus (-) signs. See Zoom Shortcut Menu (page 1591) for a description of the options that are available while zooming in real time.



The current drawing area is used to determine the zooming factor. ZOOM uses half of the window height to move to a zoom factor of 100%. Holding down the pick button at the midpoint of the window and moving vertically to the top of the window zooms in to 100%. Conversely, holding the pick button down at the midpoint of the window and moving vertically to the bottom of the window zooms out by 100%.

**NOTE** If you place the cursor at the bottom of the window, hold down the pick button, and move vertically to the top of the window, the zoom-in factor is 200%.

When you have reached the zoom-in limit, the plus sign in the cursor disappears, indicating that you can no longer zoom in. When you have reached the zoom-out limit, the minus sign in the cursor disappears, indicating that you can no longer zoom out.

When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue to zoom the display from that location.

To exit zooming at the new position, press ENTER or ESC.

# Zoom Shortcut Menu

## Quick Reference

When the ZOOM command is active, you can exit ZOOM or switch to *PAN* or *3DORBIT* using the options on the Zoom shortcut menu. To access the Zoom shortcut menu, right-click in the drawing area while ZOOM is active.

**Exit** Cancels ZOOM or PAN.

**Pan** Switches to PAN.

**Zoom** Switches to ZOOM in real time.

**3D Orbit** Switches to 3DORBIT.

**Zoom Window** Zooms to display an area specified by a rectangular window.

**Zoom Object** Zooms to display one or more selected objects as large as possible and in the center of the drawing area.

**Zoom Original** Restores the original view.

**Zoom Extents** Zooms to display the drawing extents.

# Command Modifiers

You can use command modifiers to help you locate points or select objects while a command is in progress. Use the Coordinate Filter, Direct Distance Entry, From, MTP, and Tracking command modifiers at any prompt that requires point specification. Use the Selection Mode command modifiers at any prompt that requires object selection.

## In this part

# Command Modifiers

# 26

## In this chapter

# Coordinate Filters (Command Modifier)

## Quick Reference

Combines X, Y, and Z values from different points to specify a single point

*Enter* **.x**, **.y**, **.xy**, **.xz**, *or* **.yz**

At any prompt for locating a point, you can enter point filters to specify a single coordinate by extracting the *X*, *Y*, and *Z* values of several points. In the following example, the start point for the line has a coordinate constructed from the *X* value of the midpoint of the first object you select, with the *Y* and *Z* values of the midpoint of the second object you select.

Command: **line**
Specify first point: **.x**
of **mid**
of *Select an object*
of (need YZ) **mid**
of *Select another object*
To point: *Specify a point*

# Direct Distance Entry (Command Modifier)

## Quick Reference

Locates the next point at a specified distance in the direction of your cursor

*At a prompt to locate a point, enter a numeric value*

With direct distance entry, you can quickly specify a point relative to the last point you entered. At any prompt for a point location, you move the cursor first to specify the direction, and then enter a numeric distance.

In the following example, the second point for the line will be located 5 units toward the direction of the cursor. The direct distance that you enter is measured along the path from the last point to the current location of the cursor. This feature is usually used with Ortho or Snap mode turned on.

Command: **line**
Specify first point: *Specify a point*
Specify next point: *Move the cursor in the desired direction and enter* **5**

**NOTE** The direct distance entry method is not available while you are using temporary override keys for Ortho, object snap tracking, or polar tracking.

# FROM (Command Modifier)

## Quick Reference

Locates a point offset from a reference point within a command

Object Snap
*At a prompt to locate a point, enter* **from**
Base Point: *Specify a point to use as a base point*
of <offset>: *Enter a relative offset*

At a prompt for locating a point, enter **from**, and then enter a temporary reference or base point from which you can specify an offset to locate the next point. Enter the offset location from this base point as a relative coordinate, or use direct distance entry (page 1596).

**NOTE** You cannot use this method during dragging in commands such as MOVE and COPY. Specifying an absolute coordinate, either by keyboard entry or with a pointing device, cancels the FROM command.

# MTP (Command Modifier)

## Quick Reference

Locates the midpoint between two points

*At a prompt to locate a point, enter* **mtp**

The midpoint between two points (MTP) command modifier can be used with running object snaps or object snap overrides. (*OSNAP* command)

At any Command prompt to locate a point, enter **mtp** or **m2p**.

First point of mid: *Specify a point*
Second point of mid: *Specify a second point*

# TRACKING (Command Modifier)

## Quick Reference

Locates a point from a series of temporary points

*At a prompt to locate a point, enter* **tracking**

Tracking turns on Ortho mode and can be used with Snap mode.

At any prompt to locate a point, enter **tracking**, **track**, or **tk**.

First tracking point: *Specify a location or distance*
Next point (Press ENTER to end tracking): *Specify a second location or distance*

Tracking specifies a series of temporary points, each offset from the previous one. Thus, you specify a new point location from a series of directions and distances. To determine the location of each temporary point, you can use direct distance entry (page 1596). First move the cursor to specify the direction, and then enter a numeric distance.

Alternatively, tracking can establish a new point by combining the *X* and *Y* values of two specified points. You can specify these two points in any order, depending on the cursor direction after the first point. Direct distance entry specifies a distance in the direction of the current location of your cursor. Coordinate filters (page 1596) combine *X*, *Y*, and *Z* values from different points into a single point. Relative coordinate entry locates a point relative to the last point entered.

You can also access tracking by holding down SHIFT and right-clicking to display the object snap shortcut menu.

# Object Snaps (Command Modifier)

## Quick Reference

Specifies a precise point at a location on an object

*Within a command, at a prompt to locate a point, specify an object snap*

When you specify an object snap, the cursor snaps to the specified point on an object closest to the center of the cursor. By default, a marker and a tooltip are displayed when you move the cursor over the object snap location on an object.

### Specify a Single Object Snap

If you specify a single object snap, it stays in effect only for the next point you specify.

You can specify an object snap with any of the following methods:

- Enter a object snap by typing its name. To see a list of valid object snaps, refer to the OSNAP (page 996) command or the Drafting Settings Dialog Box (page 488).
- Click an object snap from the Object Snap toolbar.
- Click an object snap from the Object Snap shortcut menu. You can diisplay this shortcut menu by pressing SHIFT while you right-click.

### Use Running Object Snaps

Using the *OSNAP* or *DSETTINGS* commands, you can specify a set of *running object snaps*. Running object snaps are one or more object snaps that remain in effect as you work.

- To turn running object snaps on and off, click the OSNAP button on the status bar or press F3.
- Press TAB to cycle through the object snap possibilities before you specify the point.
- To turn off running object snaps for the next point only, specify the None object snap.

# Selection Modes (Command Modifier)

## Quick Reference

Controls how you create selection sets

*At a prompt to select objects, enter one of the valid selection modes described in the SELECT command*

The selection mode you enter remains active only for the current Select Objects prompt.

# System Variables

# 27

AutoCAD® stores the values for its operating environment and some of its commands in system variables. You can examine any system variable and change any writable system variable directly at the command prompt by entering the system variable name or by using the SETVAR command or the AutoLISP® `getvar` and `setvar` functions. Many system variables are also accessible through dialog box options.

To access a list of system variables, on the Contents tab of the Help window, click the + sign next to System Variables.

## In this chapter

- W System Variables
- X System Variables
- Z System Variables

# 3D System Variables

## 3DDWFPREC

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Controls the precision of 3D DWF publishing.

This system variable has a range from 1 to 6. Higher settings result in finer precision.

| 3DDWFPREC Value | Deviation Value |
|---|---|
| 1 | 1 |
| 2 | 0.5 |
| 3 | 0.2 |
| 4 | 0.1 |
| 5 | 0.01 |
| 6 | 0.001 |

**NOTE** Setting a 3DDWFPREC value of 5 or 6 will create very large 3D DWF files or sheets in a multi-sheet DWF.

## 3DCONVERSIONMODE

### Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**1
Used to convert material and light definitions to the current product release

| 3DCONVERSIONMODE Value | Deviation Value |
|---|---|
| 0 | No material or lighting conversion takes place when the drawing is opened |
| 1 | Material and lighting conversion takes place automatically |
| 2 | You are prompted to convert any materials or lighting. |

# 3DSELECTIONMODE

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:**1
Controls the selection precedence of visually overlapping objects when using 3D visual styles

3DSELECTIONMODE has no effect when selecting 3D solids if they are displayed as 2D or 3D wireframes.

This system variable is intended to support legacy selection behavior for the current release only. It will be removed in the future.

| | |
|---|---|
| 0 | Use legacy 3D selection precedence |
| 1 | Use line-of-sight 3D selection precedence |

# A System Variables

## ACADLSPASDOC

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether the acad.lsp file is loaded into every drawing or just the first drawing opened in a session.

| | |
|---|---|
| 0 | Loads *acad.lsp* into just the first drawing opened in a session |
| 1 | Loads *acad.lsp* into every drawing opened |

## ACADPREFIX

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**pathname
Stores the directory path, if any, specified by the ACAD environment variable, with path separators appended if necessary.

## ACADVER

### Quick Reference

(Read-only)
**Type:** String

**Saved in:**Not-saved
Stores the AutoCAD version number. This variable differs from the DXF file $ACADVER header variable, which contains the drawing database level number.

# ACISOUTVER

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**70
Controls the ACIS version of SAT files created using the ACISOUT command. ACISOUT only supports a value of 15 through 18, 20, 21, 30, 31, 40, 50, 60, and 70.

# ADCSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**varies
Indicates whether the DesignCenter & trade window is open or closed. For developers who need to determine status through AutoLISP .

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# AFLAGS

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 16
Sets options for attributes. The value is the sum of the following:

| | |
|---|---|
| 0 | No attribute mode selected |
| 1 | Invisible |
| 2 | Constant |
| 4 | Verify |
| 8 | Preset |
| 16 | Lock position in block |
| 32 | Multiple lines |

# ANGBASE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the base angle to 0 with respect to the current UCS.

# ANGDIR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the direction of positive angles. Angle values are measured from angle 0 relative to the orientation of the current UCS.

| | |
|---|---|
| 0 | Counterclockwise |
| 1 | Clockwise |

# ANNOALLVISIBLE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Hides or displays objects that do not support the current

| | |
|---|---|
| 0 | Only annotative objects that support the current are displayed |
| 1 | All annotative objects are displayed |

The ANNOALLVISIBLE setting is saved individually for model space and each layout.

**NOTE** When ANNOALLVISIBLE is set to 1, annotative objects that support more than one scale will only display one .

# ANNOAUTOSCALE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** -4
Updates objects to support the when the annotation scale is changed

When the value is negative, the autoscale functionality is turned off, but the settings are maintained:

| | |
|---|---|
| 1 | Adds the newly set annotation scale to annotative objects that support the current scale except for those on layers that are turned off, frozen, locked or that are set to Viewport > Freeze. |
| -1 | ANNOAUTOSCALE is turned off, but when turned back on is set to 1. |
| 2 | Adds the newly set annotation scale to annotative objects that support the current scale except for those on layers that are turned off, frozen, or that are set to Viewport > Freeze. |
| -2 | ANNOAUTOSCALE is turned off, but when turned back on is set to 2. |
| 3 | Adds the newly set annotation scale to annotative objects that support the current scale except for those on layers that are locked. |
| -3 | ANNOAUTOSCALE is turned off, but when turned back on is set to 3. |
| 4 | Adds the newly set annotation scale to all annotative objects that support the current scale. |

| | |
|---|---|
| -4 | ANNOAUTOSCALE is turned off, but when turned back on is set to 4. |

# ANNOTATIVEDWG

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Specifies whether or not the drawing will behave as an block when inserted into another drawing

| | |
|---|---|
| 0 | Nonannotative |
| 1 | Annotative |

**NOTE** The ANNOTATIVEDWG system variable becomes read-only if the drawing contains annotative objects.

# APBOX

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Turns the display of the AutoSnap™ aperture box on or off. The aperture box is displayed in the center of the crosshairs when you snap to an object.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# APERTURE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 10
Sets the display size for the object snap target box, in pixels. This system variable has the same name as a command. Use the SETVAR command to access this system variable.

Enter a value (1-50). The higher the number, the larger the target box. You can also change this setting in the Options dialog box, Drafting tab (page 972).

APERTURE controls the object snap target box, not the pickbox displayed at the Select Objects prompt. The object selection pickbox is controlled by the *PICKBOX* system variable.

# APSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Stores a value that indicates whether the Block Authoring Palettes window in the Block Editor is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# AREA

### Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Not-saved
**Initial value:** None
Stores the last area computed by the AREA command. This system variable has the same name as a command. Use the SETVAR command to access this system variable.

# ATTDIA

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls whether the INSERT command uses a dialog box for attribute value entry.

| | |
|---|---|
| 0 | Issues command prompts |
| 1 | Uses a dialog box |

# ATTIPE

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0

Controls the display of the in-place editor used to create multiline attributes.

| | |
|---|---|
| 0 | Displays a limited in-place editor that includes only a few formatting options |
| 1 | Displays the full in-place editor with all formatting options |

Use the limited in-place editor when creating multiline attributes for best compatibility with releases prior to AutoCAD 2008.

# ATTMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls display of attributes.

| | |
|---|---|
| 0 | Off: Makes all attributes invisible |
| 1 | Normal: Retains current visibility of each attribute; visible attributes are displayed; invisible attributes are not |
| 2 | On: Makes all attributes visible |

# ATTMULTI

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1

Controls whether multiline attributes can be created.

| | |
|---|---|
| 0 | Turns off all access methods for creating multiline attributes. They can still be viewed and edited. |
| 1 | Turns on all access methods for creating multiline attributes. |

# ATTREQ

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether INSERT uses default attribute settings during insertion of blocks.

| | |
|---|---|
| 0 | Assumes the defaults for the values of all attributes |
| 1 | Turns on prompts or a dialog box for attribute values, as specified by *ATTDIA* |

# AUDITCTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls whether AUDIT creates an audit report (ADT) file.

| | |
|---|---|
| 0 | Prevents writing of ADT files |
| 1 | Writes ADT files |

# AUNITS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets units for angles.

| | |
|---|---|
| 0 | Decimal degrees |
| 1 | Degrees/minutes/seconds |
| 2 | Gradians |
| 3 | Radians |
| 4 | Surveyor's units |

# AUPREC

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the number of decimal places for all read-only angular units displayed on the status line, and for all editable angular units whose precision is less than or equal to the current AUPREC value. For editable angular units whose precision is greater than the current AUPREC value, the true precision is displayed. AUPREC does not affect the display precision of dimension text (see DIMSTYLE).

# AUTODWFPUBLISH

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:** 0
Controls whether the AutoPublish feature is on or off

| | |
|---|---|
| 0 | Turns off the AutoPublish feature |
| 1 | Turns on the AutoPublish feature |

# AUTOSNAP

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 63
Controls the display of the AutoSnap marker, tooltip, and magnet. Also turns on polar and object snap tracking, and controls the display of polar tracking, object snap tracking, and Ortho mode tooltips. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Turns off the AutoSnap marker, tooltips, and magnet. Also turns off polar tracking, object snap tracking, and tooltips for polar tracking, object snap tracking, and Ortho mode |
| 1 | Turns on the AutoSnap marker |
| 2 | Turns on the AutoSnap tooltips |
| 4 | Turns on the AutoSnap magnet |
| 8 | Turns on polar tracking |

| | |
|---|---|
| 16 | Turns on object snap tracking |
| 32 | Turns on tooltips for polar tracking, object snap tracking, and Ortho mode |

# B System Variables

# BACKGROUNDPLOT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2

Controls whether background plotting is turned on or off for plotting and publishing. By default, background plotting is turned off for plotting and on for publishing.

| Value | PLOT | PUBLISH |
|---|---|---|
| 0 | Foreground | Foreground |
| 1 | Background | Foreground |
| 2 | Foreground | Background |
| 3 | Background | Background |

When -PLOT, PLOT, -PUBLISH, and PUBLISH are used in a script (SCR file), the BACKGROUNDPLOT system variable value is ignored, and -PLOT, PLOT, -PUBLISH, and PUBLISH are processed in the foreground.

# BACKZ

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** None
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

Meaningful only if clipping is specified in *CAMERA*, *DVIEW*, or *3DCLIP*. If there are several cameras, the value is the last back clipping plane that you set current. The distance of the back clipping plane from the camera point can be found by subtracting BACKZ from the camera-to-target distance.

# BACTIONCOLOR

### Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** 7
Sets the text color of actions in the Block Editor. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255.

Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

# BDEPENDENCYHIGHLIGHT

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1

Controls whether or not dependent objects are dependency highlighted when a parameter, action, or grip is selected in the Block Editor.

| | |
|---|---|
| 0 | Specifies that dependent objects are not highlighted |
| 1 | Specifies that dependent objects are highlighted |

Dependency highlighting displays objects with a halo effect.

# BGRIPOBJCOLOR

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**141
Sets the color of grips in the Block Editor. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255.

Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

# BGRIPOBJSIZE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**8
Sets the display size of custom grips in the Block Editor relative to the screen display. Valid values include an integer from 1 to 255.

Use the *REGEN* command to update the display size of custom grips in the Block Editor.

# BINDTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls how xref names are handled when binding xrefs or editing xrefs in place.

| | |
|---|---|
| 0 | Traditional binding behavior ("xref1\|one" becomes "xref$0$one") |
| 1 | Insert-like behavior ("xref1\|one" becomes "one") |

# BLIPMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether marker blips are visible. This system variable has the same name as a command. Use the *SETVAR* command to access this system variable.

| | |
|---|---|
| 0 | Turns off marker blips |
| 1 | Turns on marker blips |

# BLOCKEDITLOCK

## Quick Reference

**Type:** Integer
**Saved in:** Registry

**Initial value:**0
Disallows opening of the Block Editor and editing of dynamic block definitions.

When BLOCKEDITLOCK is set to 1, double-clicking a dynamic block in a drawing opens the Properties palette. If the dynamic block contains attributes, double-clicking the block reference opens theEnhanced Attribute Editor. Double click actions can be customized using the Create a Double Click Action.

| | |
|---|---|
| 0 | Specifies that the Block Editor can be opened |
| 1 | Specifies that the Block Editor cannot be opened |

# BLOCKEDITOR

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Reflects whether or not the Block Editor is open.

| | |
|---|---|
| 0 | Indicates that the Block Editor is not open |
| 1 | Indicates that the Block Editor is open |

# BPARAMETERCOLOR

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**7
Sets the color of parameters in the Block Editor. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255.

Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

# BPARAMETERFONT

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** Simplex.shx
Sets the font used for parameters and actions in the Block Editor.

You can specify either a True Type font or a SHX font (for example, **Verdana** or **Verdana.ttf**). You must add the *.shx* extension to specify an AutoCAD SHX font. When specifying an Asian Big Font, use the following naming convention: an SHX file followed by a comma (,), followed by the Big Font file name (for example, **Simplex.shx,Bigfont.shx**).

# BPARAMETERSIZE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 12
Sets the size of parameter text and features in the Block Editor relative to the screen display. Valid values include an integer from 1 to 255.

# BTMARKDISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1

Controls whether or not value set markers are displayed for dynamic block references.

| | |
|---|---|
| 0 | Specifies that value set markers are not displayed |
| 1 | Specifies that value set markers are displayed |

# BVMODE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls how objects that are made invisible for the current visibility state are displayed in the Block Editor.

| | |
|---|---|
| 0 | Specifies that hidden objects are not visible |
| 1 | Specifies that hidden objects are visible but dimmed |

# C System Variables

# CALCINPUT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

| | |
|---|---|
| 0 | Expressions are not evaluated |

| | |
|---|---|
| 1 | Expressions are evaluated after you press ALT+ENTER |

# CAMERADISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Turns the display of camera objects on or off. The value changes to 1 (to display cameras) when you use the CAMERA command.

| | |
|---|---|
| 0 | Camera glyphs are not displayed |
| 1 | Camera glyphs are displayed |

# CAMERAHEIGHT

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0
Specifies the default height for new camera objects. The height is expressed in current drawing units.

# CANNOSCALE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**1:1
Sets the name of the current for the current space

You can only enter a named scale that exists in the drawing's named scale list.

# CANNOSCALEVALUE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:**1
Returns the value of the current

# CDATE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:**Not-saved
Stores the current date and time in decimal format.

The date and time displays in a decimal format starting with the year. After the decimal point, CDATE displays the time using a 24-hour clock. For example, the ninth day of February in the year 2006 at 3:05 pm displays as 20060209.150500.

# CECOLOR

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**BYLAYER
Sets the color of new objects. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255.

Valid values for true colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB. The true color setting is entered as follows:

RGB:000,000,000.

# CELTSCALE

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 1.0000
Sets the current object linetype scaling factor. Sets the linetype scaling for new objects relative to the LTSCALE command setting. A line created with CELTSCALE = 2 in a drawing with LTSCALE set to 0.5 would appear the same as a line created with CELTSCALE = 1 in a drawing with LTSCALE = 1.

# CELTYPE

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** BYLAYER
Sets the linetype of new objects.

# CELWEIGHT

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** -1
Sets the lineweight of new objects.

| | |
|---|---|
| -1 | Sets the lineweight to "BYLAYER." |
| -2 | Sets the lineweight to "BYBLOCK." |
| -3 | Sets the lineweight to "DEFAULT." "DEFAULT" is controlled by the *LWDEFAULT* system variable. |

Other valid values entered in hundredths of millimeters include 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.

All values must be entered in hundredths of millimeters. (Multiply a value by 2540 to convert values from inches to hundredths of millimeters.)

# CENTERMT

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:**0
Controls how grips stretch multiline text that is centered horizontally. CENTERMT does not apply to stretching multiline text by using the ruler in the In-Place Text Editor.

| | |
|---|---|
| 0 | When you move a corner grip in centered multiline text, the center grip moves in the same direction, and the grip on the opposite side remains in place |
| 1 | When you move a corner grip in centered multiline text, the center grip stays in place, and both sets of side grips move in the direction of the stretch |

# CHAMFERA

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the first chamfer distance when CHAMMODE is set to 0.

# CHAMFERB

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the second chamfer distance when CHAMMODE is set to 0.

# CHAMFERC

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the chamfer length when CHAMMODE is set to 1.

# CHAMFERD

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the chamfer angle when CHAMMODE is set to 1.

# CHAMMODE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Sets the input method for CHAMFER.

| | |
|---|---|
| 0 | Requires two chamfer distances |
| 1 | Requires a chamfer length and an angle |

# CIRCLERAD

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:**0.0000
Sets the default circle radius. A zero indicates no default.

# CLAYER

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**0
Sets the current layer.

# CLEANSCREENSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores a value that indicates whether the clean screen state is on or off.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# CLISTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Stores a value that indicates whether the command window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# CMATERIAL

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**BYLAYER

Sets the material of new objects. Valid values are BYLAYER, BYBLOCK, and the name of a material in the drawing.

# CMDACTIVE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** None
Indicates whether an ordinary command, transparent command, script, or dialog box is active. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 1 | Ordinary command is active |
| 2 | Ordinary command and a transparent command are active |
| 4 | Script is active |
| 8 | Dialog box is active |
| 16 | DDE is active |
| 32 | AutoLISP is active (only visible to an ObjectARX-defined command) |
| 64 | ObjectARX command is active |

# CMDDIA

## Quick Reference

**Type:** Integer
**Saved in:** Registry

**Initial value:**1
Controls the display of the In-Place Text Editor for the QLEADER command.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# CMDECHO

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Controls whether prompts and input are echoed during the AutoLISP command function.

| | |
|---|---|
| 0 | Turns off echoing |
| 1 | Turns on echoing |

# CMDINPUTHISTORYMAX

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**20
Sets the maximum number of previous input values that are stored for a prompt in a command. Display of the history of user input is controlled by the INPUTHISTORYMODE (page 1725) system variable.

# CMDNAMES

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:** None
Displays the names of the active and transparent commands. For example, LINE'ZOOM indicates that the ZOOM command is being used transparently during the LINE command.

This variable is designed for use with programming interfaces such as AutoLISP, DIESEL, and ActiveX Automation.

The following is a simple example that demonstrates how to use DIESEL to display the current command at the status line.

Command: modemacro
New value for MODEMACRO, or . for none <"">: $(getvar, cmdnames)

For additional information, see "Introduction to Programming Interfaces" in the *Customization Guide*.

# CMLEADERSTYLE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** STANDARD
Sets the name of the current multileader style

# CMLJUST

## Quick Reference

**Type:** Integer
**Saved in:** Drawing

**Initial value:**0
Specifies multiline justification.

| | |
|---|---|
| 0 | Top |
| 1 | Zero (Middle) |
| 2 | Bottom |

# CMLSCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**1.0000 (imperial) or 20.0000 (metric)
Controls the overall width of a multiline. A scale factor of 2.0 produces a multiline twice as wide as the style definition. A zero scale factor collapses the multiline into a single line. A negative scale factor flips the order of the offset lines (that is, the smallest or most negative is placed on top when the multiline is drawn from left to right).

# CMLSTYLE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**STANDARD
Sets the multiline style that governs the appearance of the multiline.

# COMPASS

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls whether the 3D compass is on or off in the current viewport.

| | |
|---|---|
| 0 | Turns off the 3D compass |
| 1 | Turns on the 3D compass |

# COORDS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the format and update frequency of coordinates on the status line.

| | |
|---|---|
| 0 | The absolute coordinates of your pointing device are updated only when you specify points |
| 1 | The absolute coordinates of your pointing device are updated continuously |
| 2 | The absolute coordinates of your pointing device are updated continuously except when a point, distance, or angle is requested. In that case, relative polar coordinates are displayed instead of X and Y. The Z values always display as absolute coordinates. |

# COPYMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether the COPY command repeats automatically.

| | |
|---|---|
| 0 | Sets the COPY command to repeat automatically |
| 1 | Sets the COPY command to create a single copy |

# CPLOTSTYLE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**Varies
Controls the current plot style for new objects. If the current drawing you are working in is in color-dependent mode (PSTYLEPOLICY is set to 1), CPLOTSTYLE is read-only and has a value of BYCOLOR. If the current drawing you are working in is in named plot styles mode (PSTYLEPOLICY is set to 0), CPLOTSTYLE can be set to the following values (BYLAYER is the default):

"BYLAYER"

"BYBLOCK"

"NORMAL"

"USER DEFINED"

To convert the current drawing to use named or color-dependent plot styles, use *CONVERTPSTYLES*.

# CPROFILE

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**<<Unnamed Profile>>
Displays the name of the current profile. For more information on profiles, see the OPTIONS (page 946) command.

# CROSSINGAREACOLOR

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**3 (green)
Controls the color of the selection area during crossing selection. The valid range is 1 to 255. The SELECTIONAREA system variable must be on.

# CSHADOW

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the shadow display property for a 3D object. To be visible, shadows must be turned on in the visual style that is applied to the viewport.

| | |
|---|---|
| 0 | Casts and receives shadows |
| 1 | Casts shadows |
| 2 | Receives shadows |

3 Ignores shadows

**NOTE** To display full shadows, hardware acceleration is required. When Geometry Acceleration is off, full shadows cannot be displayed. (To access these settings, enter **3dconfig** at the command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.)

# CTAB

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** Varies
Returns the name of the current (model or layout) tab in the drawing. Provides a means for the user to determine which tab is active.

# CTABLESTYLE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** STANDARD
Sets the name of the current table style

# CURSORSIZE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 5
Determines the size of the crosshairs as a percentage of the screen size. Valid settings range from 1 to 100 percent. When set to 100, the crosshairs are

full-screen and the ends of the crosshairs are never visible. When less than 100, the ends of the crosshairs may be visible when the cursor is moved to one edge of the screen.

## CVPORT

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Displays the identification number of the current viewport. You can change this value, which changes the current viewport, if the following conditions are met:

- The identification number you specify is that of an active viewport.
- A command in progress has not locked cursor movement to that viewport.
- Tablet mode is off.

# D System Variables

## DASHBOARDSTATE

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**Varies
Indicates whether the Dashboard window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# DATALINKNOTIFY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Controls the notification for updated or missing data links

| | |
|---|---|
| 0 | Disables data link update notification. |
| 1 | Enables data link notification. Notifies you that external data is linked to the current drawing by displaying the data link icon in the lower-right corner of the application window (the notification area of the status bar tray). Will not display balloon message notifications. |
| 2 | Enables data link notification and balloon message notifications. Notifies you that external data is linked to the current drawing as in 1 above. Also displays a data link file change notification in the area of the drawing where the data link requires updating. |

# DATE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Not-saved
**Initial value:**Varies
Stores the current date and time in Modified Julian Date format. This value is represented as a Modified Julian Date (MJD), which is the Julian day number and decimal fraction of a day in the format :

<Julian day number>.<Decimal fraction of a day>

The Modified Julian Date, conventionally called UT1, is a worldwide scientific standard that assigns day numbers beginning at an essentially arbitrary date

and time of 12:00 a.m. on 1 January 4713 B.C. (B.C.E.). With this system, 4 July 1997 at 2:29:58 p.m. corresponds to 2450634.60387736, and 1 January 1998 at 12:00 noon corresponds to 2450815.50000000.

You can compute differences in date and time by subtracting the numbers returned by DATE. To extract the seconds since midnight from the value returned by DATE, use AutoLISP expressions:

```
(setq s (getvar "DATE"))
(setq seconds (* 86400.0 (- s (fix s))))
```

Because your computer clock provides the date and time, the DATE system variable returns a true Julian date only if the system clock is set to UTC/Zulu (Greenwich Mean Time). *TDCREATE* and *TDUPDATE* have the same format as DATE, but their values represent the creation time and last update time of the current drawing.

# DBCSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores the state of the dbConnect Manager is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# DBLCLKEDIT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1

Controls the double click editing behavior in the drawing area. Double click actions can be customized using the Customize User Interface (CUI) editor. The system variable can accept the values of On and Off in place of 1 and 0.

| | |
|---|---|
| 0 | Disabled |
| 1 | Enabled |

# DBMOD

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Indicates the drawing modification status. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 1 | Object database modified |
| 4 | Database variable modified |
| 8 | Window modified |
| 16 | View modified |
| 32 | Field modified |

The DBMOD value is reset to 0 when you save the drawing.

# DCTCUST

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** pathname
Displays the path and file name of the current custom spelling dictionary.

# DCTMAIN

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** Varies by country/region
Displays the three letter keyword for the current main spelling dictionary.

You can specify a default main spelling dictionary using the *SETVAR* command. When prompted for a new value for DCTMAIN, you can enter one of the keywords below. Depending on the language version of the program, not all dictionaries may be available.

| Keyword | Language name |
|---|---|
| enu | American English |
| eng | British English (ise) |
| enc | Canadian English |
| cat | Catalan |
| csy | Czech |
| dan | Danish |

| | |
|---|---|
| nld | Dutch (primary) |
| fin | Finnish |
| fra | French (accented capitals) |
| frc | French (unaccented capitals) |
| deu | German (post-reform) |
| deo | German (pre-reform) |
| ita | Italian |
| nor | Norwegian (Bokmal) |
| ptb | Portuguese (Brazilian) |
| ptg | Portuguese (Iberian) |
| rus | Russian |
| esp | Spanish |
| sve | Swedish |

# DEFAULTLIGHTING

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1

Controls default lighting in the current viewport. Default lighting is provided by a set of distant lights that follow the view direction. When default lighting

is on, the sun and other lights do not cast light, even if they are turned on. The setting of this system variable is viewport-specific.

| | |
|---|---|
| 0 | Auto: default lighting is automatically turned off when point lights, spotlights, distant lights, or the sun are on |
| 1 | On |

# DEFAULTLIGHTINGTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Specifies the type of default lighting. The default lighting provided in AutoCAD 2006 and earlier releases used one distant light. The new default lighting uses two distant lights to illuminate more of the model and also adjusts ambient light. The setting of this system variable is viewport-specific.

| | |
|---|---|
| 0 | Old type of default lighting |
| 1 | New type of default lighting |

# DEFLPLSTYLE

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** Varies
Specifies the default plot style for all layers in a drawingwhen opening a drawing that was created in a release prior to AutoCAD 2000, or for Layer 0 when creating a new drawing from scratch without using a drawing template.

When the drawing is opened and PSTYLEPOLICY (page 1794) is set to 1 (color-dependent plot style mode), DEFLPLSTYLE is read-only and has a value

of “BYCOLOR.” If PSTYLEPOLICY is set to 0 (named plot style mode) when the drawing is opened, DEFLPLSTYLE is writable and has a default value of “NORMAL.”

To convert the current drawing to use named or color-dependent plot styles, use *CONVERTPSTYLES*.

# DEFPLSTYLE

## Quick Reference

**Type:** String
**Saved in:** Registry
Specifies the default plot style for new objects in a drawing when opening a drawing that was created in a release prior to AutoCAD 2000, or when creating a new drawing from scratch without using a drawing template.

When the drawing is opened and PSTYLEPOLICY (page 1794) is set to 1(color-dependent plot style mode), DEFPLSTYLE is read-only and has a value of “BYCOLOR.” If PSTYLEPOLICY is set to 0 (named plot style mode), DEFPLSTYLE is writable and has a default value of “BYLAYER.”

To convert the current drawing to use named or color-dependent plot styles, use *CONVERTPSTYLES*.

# DELOBJ

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether geometry used to create 3D objects is retained or deleted.

| | |
|---|---|
| 0 | All defining geometry is retained. |

| | |
|---|---|
| 1 | Profile curves are deleted, including those used with the *EXTRUDE*, *SWEEP*, *REVOLVE*, and *LOFT* commands. Cross sections used with the LOFT command are also deleted. |
| 2 | All defining geometry is deleted, including paths and guide curves used with the SWEEP and LOFT commands. |
| -1 | Prompt to delete profile curves, including those used with the EXTRUDE, SWEEP, REVOLVE, and LOFT commands. Cross sections used with the LOFT command are also deleted. |
| -2 | Prompt to delete all defining geometry, including paths and guide curves used with the SWEEP and LOFT commands. |

# DEMANDLOAD

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** Initial Value: 3

Specifies if and when to demand-load certain applications. If you set this system variable to 0, third-party applications and some AutoCAD commands cannot function.

| | |
|---|---|
| 0 | Turns off demand-loading. |
| 1 | Demand-loads the source application when you open a drawing that contains custom objects. This setting does not demand-load the application when you invoke one of the application's commands. |
| 2 | Demand-loads the source application when you invoke one of the application's commands. This setting does not |

| | |
|---|---|
| | demand-load the application when you open a drawing that contains custom objects. |
| 3 | Demand-loads the source application when you open a drawing that contains custom objects or when you invoke one of the application's commands |

# DGNFRAME

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Determines whether DGN underlay frames are visible or plotted in the current drawing.

| | |
|---|---|
| 0 | The DGN underlay frame is not displayed or plotted |
| 1 | The DGN underlay frame is displayed and plotted |
| 2 | The DGN underlay frame is displayed but not plotted |

# DGNOSNAP

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls object snapping for geometry in DGN underlays.

| | |
|---|---|
| 0 | Turns off object snapping for geometry in all DGN underlay attachments in the drawing |

| | |
|---|---|
| 1 | Turns on object snapping for geometry in all DGN underlay attachments in the drawing |

# DIASTAT

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**None
Stores the exit method of the most recently used dialog box.

| | |
|---|---|
| 0 | Cancel |
| 1 | OK |

**NOTE** The DIASTAT system variable does not apply to standard file selection dialog boxes, such as the Select File and the Save Drawing As dialog boxes.

# DIMADEC

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the number of precision places displayed in angular dimensions.

| | |
|---|---|
| -1 | Angular dimensions display the number of decimal places specified by *DIMDEC*. |
| 0-8 | Specifies the number of decimal places displayed in angular dimensions (independent of DIMDEC) |

# DIMALT

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls the display of alternate units in dimensions.

See also *DIMALTD*, *DIMALTF*, *DIMALTTD*, *DIMALTTZ*, *DIMALTZ*, and *DIMAPOST*.

| | |
|---|---|
| Off | Disables alternate units |
| On | Enables alternate units |

# DIMALTD

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Controls the number of decimal places in alternate units. If DIMALT is turned on, DIMALTD sets the number of digits displayed to the right of the decimal point in the alternate measurement.

# DIMALTF

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 25.4000
Controls the multiplier for alternate units. If DIMALT is turned on, DIMALTF multiplies linear dimensions by a factor to produce a value in an alternate

system of measurement. The initial value represents the number of millimeters in an inch.

# DIMALTRND

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.00
Rounds off the alternate dimension units.

# DIMALTTD

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

# DIMALTTZ

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls suppression of zeros in tolerance values.

| | |
|---|---|
| 0 | Suppresses zero feet and precisely zero inches |
| 1 | Includes zero feet and precisely zero inches |

| | |
|---|---|
| 2 | Includes zero feet and suppresses zero inches |
| 3 | Includes zero inches and suppresses zero feet |

To suppress leading or trailing zeros, add the following values to one of the preceding values:

| | |
|---|---|
| 4 | Suppresses leading zeros |
| 8 | Suppresses trailing zeros |

# DIMALTU

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Sets the units format for alternate units of all dimension substyles except Angular.

| | |
|---|---|
| 1 | Scientific |
| 2 | Decimal |
| 3 | Engineering |
| 4 | Architectural (stacked) |
| 5 | Fractional (stacked) |
| 6 | Architectural |
| 7 | Fractional |

| | |
|---|---|
| 8 | Microsoft Windows Desktop (decimal format using Control Panel settings for decimal separator and number grouping symbols) |

# DIMALTZ

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the suppression of zeros for alternate unit dimension values. DIMALTZ values 0-3 affect feet-and-inch dimensions only.

| | |
|---|---|
| 0 | Suppresses zero feet and precisely zero inches |
| 1 | Includes zero feet and precisely zero inches |
| 2 | Includes zero feet and suppresses zero inches |
| 3 | Includes zero inches and suppresses zero feet |
| 4 | Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000) |
| 8 | Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5) |
| 12 | Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5) |

# DIMANNO

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Drawing
**Initial value:** Based on current style
Indicates whether or not the current dimension style is .

| | |
|---|---|
| 0 | Nonannotative |
| 1 | Annotative |

# DIMAPOST

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** None
Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular.

For instance, if the current units are Architectural, *DIMALT* is on, *DIMALTF* is 25.4 (the number of millimeters per inch), *DIMALTD* is 2, and *DIMPOST* is set to "mm," a distance of 10 units would be displayed as 10"[254.00mm].

To turn off an established prefix or suffix (or both), set it to a single period (.).

# DIMARCSYM

## Quick Reference

**Type:** Integer
**Saved in:** Drawing

**Initial value:**0
Controls display of the arc symbol in an arc length dimension.

| | |
|---|---|
| 0 | Places arc length symbols before the dimension text |
| 1 | Places arc length symbols above the dimension text |
| 2 | Suppresses the display of arc length symbols |

# DIMASO

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:**On
Obsolete. Retained in the product to preserve the integrity of scripts. See DIMASSOC.

# DIMASSOC

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Controls the associativity of dimension objects and whether dimensions are exploded.

| | |
|---|---|
| 0 | Creates exploded dimensions. There is no association between the various elements of the dimension. The lines, arcs, arrowheads, and text of a dimension are drawn as separate objects. |
| 1 | Creates non-associative dimension objects. The elements of the dimension are formed into a single object. If one |

| | |
|---|---|
| | of the definition points of the dimension moves, the dimension is updated. |
| 2 | Creates associative dimension objects. The elements of the dimension are formed into a single object, and one or more definition points of the dimension are coupled with association points on geometric objects. If the association point on the geometric object moves, the dimension location, orientation, and value are updated. |

DIMASSOC is not stored in a dimension style.

Drawings saved in a format previous to AutoCAD 2002 retain the setting of the DIMASSOC system variable. When the drawing is reopened in AutoCAD 2002 or later, the dimension associativity setting is restored. If a legacy drawing is opened in AutoCAD 2002, the DIMASSOC system variable takes on the value of the legacy drawing's *DIMASO* system variable.

# DIMASZ

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.1800
Controls the size of dimension line and leader line arrowheads. Also controls the size of hook lines.

Multiples of the arrowhead size determine whether dimension lines and text should fit between the extension lines. DIMASZ is also used to scale arrowhead blocks if set by *DIMBLK*. DIMASZ has no effect when *DIMTSZ* is other than zero.

# DIMATFIT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing

**Initial value:**3
Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

| | |
|---|---|
| 0 | Places both text and arrows outside extension lines |
| 1 | Moves arrows first, then text |
| 2 | Moves text first, then arrows |
| 3 | Moves either text or arrows, whichever fits best |

A leader is added to moved dimension text when *DIMTMOVE* is set to 1.

# DIMAUNIT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the units format for angular dimensions.

| | |
|---|---|
| 0 | Decimal degrees |
| 1 | Degrees/minutes/seconds |
| 2 | Gradians |
| 3 | Radians |

DIMAUNIT sets this value when entered at the command prompt or when set from the Primary Units area in the Annotation dialog box.

# DIMAZIN

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Suppresses zeros for angular dimensions.

| | |
|---|---|
| 0 | Displays all leading and trailing zeros |
| 1 | Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000) |
| 2 | Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5) |
| 3 | Suppresses leading and trailing zeros (for example, 0.5000 becomes .5) |

# DIMBLK

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**None
Sets the arrowhead block displayed at the ends of dimension lines. To return to the default, closed-filled arrowhead display, enter a single period (.). Arrowhead block entries and the names used to select them in the New, Modify, and Override Dimension Style dialog boxes are shown below. You can also enter the names of user-defined arrowhead blocks.

**NOTE** blocks cannot be used as custom arrowheads for dimensions or leaders.

| | |
|---|---|
| "" | closed filled |

| | |
|---|---|
| "_DOT" | dot |
| "_DOTS-MALL" | dot small |
| "_DOTB-LANK" | dot blank |
| "_ORI-GIN" | origin indicator |
| "_ORI-GIN2" | origin indicator 2 |
| "_OPEN" | open |
| "_OPEN90" | right angle |
| "_OPEN30" | open 30 |
| "_CLOSED" | closed |
| "_SMALL" | dot small blank |
| "_NONE" | none |
| "_OB-LIQUE" | oblique |
| "_BOX-FILLED" | box filled |
| "_BOXB-LANK" | box |
| "_CLOSED-BLANK" | closed blank |

| | |
|---|---|
| "_DATUM-FILLED" | datum triangle filled |
| "_DATUMB-LANK" | datum triangle |
| "_INTEG-RAL" | integral |
| "_ARCHTICK" | architectural tick |

# DIMBLK1

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** None
Sets the arrowhead for the first end of the dimension line when DIMSAH is on. To return to the default, closed-filled arrowhead display, enter a single period (.). For a list of arrowheads, see DIMBLK.

**NOTE** blocks cannot be used as custom arrowheads for dimensions or leaders.

# DIMBLK2

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** None
Sets the arrowhead for the second end of the dimension line when DIMSAH is on. To return to the default, closed-filled arrowhead display, enter a single period (.). For a list of arrowhead entries, see DIMBLK.

**NOTE** blocks cannot be used as custom arrowheads for dimensions or leaders.

# DIMCEN

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0900
Controls drawing of circle or arc center marks and centerlines by the DIMCENTER, DIMDIAMETER, and DIMRADIUS commands. For DIMDIAMETER and DIMRADIUS, the center mark is drawn only if you place the dimension line outside the circle or arc.

| | |
|---|---|
| 0 | No center marks or lines are drawn |
| <0 | Centerlines are drawn |
| >0 | Center marks are drawn |

The absolute value specifies the size of the center mark or centerline.

The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline.

The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

# DIMCLRD

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Assigns colors to dimension lines, arrowheads, and dimension leader lines. Also controls the color of leader lines created with the LEADER command. Color numbers are displayed in the Select Color dialog box. For BYBLOCK, enter 0. For BYLAYER, enter 256.

# DIMCLRE

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Assigns colors to dimension extension lines. Color numbers are displayed in the Select Color dialog box. For BYBLOCK, enter 0. For BYLAYER, enter 256.

# DIMCLRT

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Assigns colors to dimension text. The color can be any valid color number.

# DIMDEC

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 4
Sets the number of decimal places displayed for the primary units of a dimension. The precision is based on the units or angle format you have selected.

# DIMDLE

### Quick Reference

**Type:** Real

**Saved in:** Drawing
**Initial value:** 0.0000
Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

# DIMDLI

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.3800
Controls the spacing of the dimension lines in baseline dimensions. Each dimension line is offset from the previous one by this amount, if necessary, to avoid drawing over it. Changes made with DIMDLI are not applied to existing dimensions.

# DIMDSEP

## Quick Reference

**Type:** Single-character
**Saved in:** Drawing
**Initial value:** Decimal point
Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.

When prompted, enter a single character at the command prompt. If dimension units is set to Decimal, the DIMDSEP character is used instead of the default decimal point. If DIMDSEP is set to NULL (default value, reset by entering a period), the decimal point is used as the dimension separator.

# DIMEXE

## Quick Reference

**Type:** Real

**Saved in:** Drawing
**Initial value:**0.1800
Specifies how far to extend the extension line beyond the dimension line.

# DIMEXO

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0625
Specifies how far extension lines are offset from origin points. With fixed-length extension lines, this value determines the minimum offset.

# DIMFIT

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**3
Obsolete, use DIMATFIT and DIMTMOVE instead. DIMFIT is replaced by DIMATFIT and DIMTMOVE. However, if DIMFIT is set to 0 - 3, then DIMATFIT is also set to 0 - 3 and DIMTMOVE is set to 0. If DIMFIT is set to 4 or 5, then DIMATFIT is set to 3 and DIMTMOVE is set to 1 or 2 respectively.

# DIMFRAC

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0

Sets the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional).

| | |
|---|---|
| 0 | Horizontal stacking |
| 1 | Diagonal stacking |
| 2 | Not stacked (for example, 1/2) |

# DIMFXL

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 1.0000
Sets the total length of the extension lines starting from the dimension line toward the dimension origin. The length is set in drawing units.

# DIMFXLON

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls whether extension lines are set to a fixed length. When DIMFXLON is on, extension lines are set to the length specified by DIMFXL.

# DIMGAP

## Quick Reference

**Type:** Real
**Saved in:** Drawing

**Initial value:**0.0900
Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text. Also sets the gap between annotation and a hook line created with the LEADER command. If you enter a negative value, DIMGAP places a box around the dimension text.

DIMGAP is also used as the minimum length for pieces of the dimension line. When the default position for the dimension text is calculated, text is positioned inside the extension lines only if doing so breaks the dimension lines into two segments at least as long as DIMGAP. Text placed above or below the dimension line is moved inside only if there is room for the arrowheads, dimension text, and a margin between them at least as large as DIMGAP: 2 * (*DIMASZ* + DIMGAP).

# DIMJOGANG

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**45 (90 for metric)
Determines the angle of the transverse segment of the dimension line in a jogged radius dimension. Jogged radius dimensions are often created when the center point is located off the page.

# DIMJUST

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the horizontal positioning of dimension text.

| | |
|---|---|
| 0 | Positions the text above the dimension line and center-justifies it between the extension lines |
| 1 | Positions the text next to the first extension line |

| | |
|---|---|
| 2 | Positions the text next to the second extension line |
| 3 | Positions the text above and aligned with the first extension line |
| 4 | Positions the text above and aligned with the second extension line |

# DIMLDRBLK

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** None
Specifies the arrow type for leaders. To return to the default, closed-filled arrowhead display, enter a single period (.). For a list of arrowhead entries, see DIMBLK.

**NOTE** blocks cannot be used as custom arrowheads for dimensions or leaders.

# DIMLFAC

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 1.0000
Sets a scale factor for linear dimension measurements. All linear dimension distances, including radii, diameters, and coordinates, are multiplied by DIMLFAC before being converted to dimension text. Positive values of DIMLFAC are applied to dimensions in both model space and paper space; negative values are applied to paper space only.

DIMLFAC applies primarily to nonassociative dimensions (DIMASSOC set 0 or 1). For nonassociative dimensions in paper space, DIMLFAC must be set individually for each layout viewport to accommodate viewport scaling.

DIMLFAC has no effect on angular dimensions, and is not applied to the values held in *DIMRND*, *DIMTM*, or *DIMTP*.

# DIMLIM

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Generates dimension limits as the default text. Setting DIMLIM to On turns DIMTOL off.

| | |
|---|---|
| Off | Dimension limits are not generated as default text |
| On | Dimension limits are generated as default text |

# DIMLTEX1

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** ""
Sets the linetype of the first extension line. The value is BYLAYER, BYBLOCK, or the name of a linetype.

# DIMLTEX2

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** ""

Sets the linetype of the second extension line. The value is BYLAYER, BYBLOCK, or the name of a linetype.

# DIMLTYPE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**""
Sets the linetype of the dimension line. The value is BYLAYER, BYBLOCK, or the name of a linetype.

# DIMLUNIT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Sets units for all dimension types except Angular.

| | |
|---|---|
| 1 | Scientific |
| 2 | Decimal |
| 3 | Engineering |
| 4 | Architectural (always displayed stacked) |
| 5 | Fractional (always displayed stacked) |
| 6 | Microsoft Windows Desktop (decimal format using Control Panel settings for decimal separator and number grouping symbols) |

# DIMLWD

## Quick Reference

**Type:** Enum
**Saved in:** Drawing
**Initial value:**-2
Assigns lineweight to dimension lines. Values are standard lineweights.

| | |
|---|---|
| -3 | Default (the *LWDEFAULT* value) |
| -2 | BYBLOCK |
| -1 | BYLAYER |

The integer represents 1/100th of a mm.

# DIMLWE

## Quick Reference

**Type:** Enum
**Saved in:** Drawing
**Initial value:**-2
Assigns lineweight to extension lines. Values are standard lineweights.

| | |
|---|---|
| -3 | Default (the *LWDEFAULT* value) |
| -2 | BYBLOCK |
| -1 | BYLAYER |

The integer represents 1/100th of a mm.

# DIMPOST

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** None
Specifies a text prefix or suffix (or both) to the dimension measurement. For example, to establish a suffix for millimeters, set DIMPOST to mm; a distance of 19.2 units would be displayed as 19.2 mm.

If tolerances are turned on, the suffix is applied to the tolerances as well as to the main dimension.

Use <> to indicate placement of the text in relation to the dimension value. For example, enter <>mm to display a 5.0 millimeter radial dimension as "5.0mm." If you entered mm <>, the dimension would be displayed as "mm 5.0." Use the <> mechanism for angular dimensions.

# DIMRND

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Rounds all dimensioning distances to the specified value. For instance, if DIMRND is set to 0.25, all distances round to the nearest 0.25 unit. If you set DIMRND to 1.0, all distances round to the nearest integer. Note that the number of digits edited after the decimal point depends on the precision set by DIMDEC. DIMRND does not apply to angular dimensions.

# DIMSAH

## Quick Reference

**Type:** Switch
**Saved in:** Drawing

**Initial value:**Off
Controls the display of dimension line arrowhead blocks.

| | |
|---|---|
| Off | Use arrowhead blocks set by *DIMBLK* |
| On | Use arrowhead blocks set by *DIMBLK1* and *DIMBLK2* |

# DIMSCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**1.0000
Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

Also affects the leader objects with the LEADER command.

| | |
|---|---|
| 0.0 | A reasonable default value is computed based on the scaling between the current model space viewport and paper space. If you are in paper space or model space and not using the paper space feature, the scale factor is 1.0. |
| >0 | A scale factor is computed that leads text sizes, arrowhead sizes, and other scaled distances to plot at their face values. |

DIMSCALE does not affect measured lengths, coordinates, or angles.

When DIMSCALE is set to 0, and the current dimension style is not , the overall dimension scale of dimensions created in paper space viewports is determined be the viewport scale.When the current dimension style is annotative (*DIMANNO* is set to 1), the DIMSCALE value is set to 0. Changes to the DIMSCALE value are ignored and the value is reset to 0.

# DIMSD1

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls suppression of the first dimension line and arrowhead.

When turned on, suppresses the display of the dimension line and arrowhead between the first extension line and the text.

| | |
|---|---|
| Off | First dimension line is not suppressed |
| On | First dimension line is suppressed |

# DIMSD2

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls suppression of the second dimension line and arrowhead.

When turned on, suppresses the display of the dimension line and arrowhead between the second extension line and the text.

| | |
|---|---|
| Off | Second dimension line is not suppressed |
| On | Second dimension line is suppressed |

# DIMSE1

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Suppresses display of the first extension line.

| | |
|---|---|
| Off | Extension line is not suppressed |
| On | Extension line is suppressed |

# DIMSE2

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Suppresses display of the second extension line.

| | |
|---|---|
| Off | Extension line is not suppressed |
| On | Extension line is suppressed |

# DIMSHO

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** On
Obsolete. Has no effect except to preserve the integrity of scripts.

# DIMSOXD

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Suppresses arrowheads if not enough space is available inside the extension lines.

| | |
|---|---|
| Off | Arrowheads are not suppressed |
| On | Arrowheads are suppressed |

If not enough space is available inside the extension lines and *DIMTIX* is on, setting DIMSOXD to On suppresses the arrowheads. If DIMTIX is off, DIMSOXD has no effect.

# DIMSTYLE

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Drawing
**Initial value:** STANDARD
Stores the name of the current dimension style. This system variable has the same name as a command. Use the SETVAR command to access this system variable. The DIMSTYLE system variable is read-only; to change the current dimension style, use the DIMSTYLE command.

# DIMTAD

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**0
Controls the vertical position of text in relation to the dimension line.

| | |
|---|---|
| 0 | Centers the dimension text between the extension lines. |
| 1 | Places the dimension text above the dimension line except when the dimension line is not horizontal and text inside the extension lines is forced horizontal (*DIMTIH* = 1). The distance from the dimension line to the baseline of the lowest line of text is the current *DIMGAP* value. |
| 2 | Places the dimension text on the side of the dimension line farthest away from the defining points. |
| 3 | Places the dimension text to conform to Japanese Industrial Standards (JIS). |

# DIMTDEC

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**4
Sets the number of decimal places to display in tolerance values for the primary units in a dimension. This system variable has no effect unless DIMTOL is set to On. The default for DIMTOL is Off.

# DIMTFAC

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**1.0000

Specifies a scale factor for the text height of fractions and tolerance values relative to the dimension text height, as set by DIMTXT. For example, if DIMTFAC is set to 1.0, the text height of fractions and tolerances is the same height as the dimension text. If DIMTFAC is set to 0.7500, the text height of fractions and tolerances is three-quarters the size of dimension text.

# DIMTFILL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the background of dimension text.

| | |
|---|---|
| 0 | No background |
| 1 | The background color of the drawing |
| 2 | The background specified by *DIMTFILLCLR* |

# DIMTFILLCLR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the color for the text background in dimensions. Color numbers are displayed in the Select Color dialog box. For BYBLOCK, enter 0. For BYLAYER, enter 256.

# DIMTIH

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** On
Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

| | |
|---|---|
| Off | Aligns text with the dimension line |
| On | Draws text horizontally |

# DIMTIX

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Draws text between extension lines.

| | |
|---|---|
| Off | Varies with the type of dimension. For linear and angular dimensions, text is placed inside the extension lines if there is sufficient room. For radius and diameter dimensions that don't fit inside the circle or arc, DIMTIX has no effect and always forces the text outside the circle or arc. |
| On | Draws dimension text between the extension lines even if it would ordinarily be placed outside those lines |

# DIMTM

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the minimum (or lower) tolerance limit for dimension text when DIMTOL or DIMLIM is on. DIMTM accepts signed values. If DIMTOL is on and DIMTP and DIMTM are set to the same value, a tolerance value is drawn.

If DIMTM and DIMTP values differ, the upper tolerance is drawn above the lower, and a plus sign is added to the DIMTP value if it is positive.

For DIMTM, the program uses the negative of the value you enter (adding a minus sign if you specify a positive number and a plus sign if you specify a negative number).

# DIMTMOVE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Sets dimension text movement rules.

| | |
|---|---|
| 0 | Moves the dimension line with dimension text |
| 1 | Adds a leader when dimension text is moved |
| 2 | Allows text to be moved freely without a leader |

# DIMTOFL

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls whether a dimension line is drawn between the extension lines even when the text is placed outside. For radius and diameter dimensions (when DIMTIX is off), draws a dimension line inside the circle or arc and places the text, arrowheads, and leader outside.

| | |
|---|---|
| Off | Does not draw dimension lines between the measured points when arrowheads are placed outside the measured points |
| On | Draws dimension lines between the measured points even when arrowheads are placed outside the measured points |

# DIMTOH

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** On
Controls the position of dimension text outside the extension lines.

| | |
|---|---|
| Off | Aligns text with the dimension line |
| On | Draws text horizontally |

# DIMTOL

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Appends tolerances to dimension text. Setting DIMTOL to on turns DIMLIM off.

# DIMTOLJ

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Sets the vertical justification for tolerance values relative to the nominal dimension text. This system variable has no effect unless DIMTOL is set to On. The default for DIMTOL is Off.

| | |
|---|---|
| 0 | Bottom |
| 1 | Middle |
| 2 | Top |

# DIMTP

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000

Sets the maximum (or upper) tolerance limit for dimension text when DIMTOL or DIMLIM is on. DIMTP accepts signed values. If DIMTOL is on and DIMTP and DIMTM are set to the same value, a tolerance value is drawn.

If DIMTM and DIMTP values differ, the upper tolerance is drawn above the lower and a plus sign is added to the DIMTP value if it is positive.

# DIMTSZ

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning.

| | |
|---|---|
| 0 | Draws arrowheads. |
| >0 | Draws oblique strokes instead of arrowheads. The size of the oblique strokes is determined by this value multiplied by the *DIMSCALE* value |

# DIMTVP

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Controls the vertical position of dimension text above or below the dimension line. The DIMTVP value is used when DIMTAD is off. The magnitude of the vertical offset of text is the product of the text height and DIMTVP. Setting DIMTVP to 1.0 is equivalent to setting DIMTAD to on. The dimension line splits to accommodate the text only if the absolute value of DIMTVP is less than 0.7.

# DIMTXSTY

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** STANDARD
Specifies the text style of the dimension.

# DIMTXT

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.1800
Specifies the height of dimension text, unless the current text style has a fixed height.

# DIMTZIN

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the suppression of zeros in tolerance values. DIMTZIN stores this value when you enter it on the command line or set it under Primary Units in the Annotation dialog box. DIMTZIN values 0-3 affect feet-and-inch dimensions only.

| | |
|---|---|
| 0 | Suppresses zero feet and precisely zero inches |
| 1 | Includes zero feet and precisely zero inches |
| 2 | Includes zero feet and suppresses zero inches |

| | |
|---|---|
| 3 | Includes zero inches and suppresses zero feet |
| 4 | Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000) |
| 8 | Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5) |
| 12 | Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5) |

# DIMUNIT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Obsolete. Retained in the product to preserve the integrity of scripts. DIMUNIT is replaced by DIMLUNIT and DIMFRAC.

# DIMUPT

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** Off
Controls options for user-positioned text.

| | |
|---|---|
| Off | Cursor controls only the dimension line location |
| On | Cursor controls both the text position and the dimension line location |

# DIMZIN

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the suppression of zeros in the primary unit value. DIMZIN stores this value when you enter it on the command line or set it under Primary Units in the Annotation dialog box. DIMZIN values 0-3 affect feet-and-inch dimensions only.

| | |
|---|---|
| 0 | Suppresses zero feet and precisely zero inches |
| 1 | Includes zero feet and precisely zero inches |
| 2 | Includes zero feet and suppresses zero inches |
| 3 | Includes zero inches and suppresses zero feet |
| 4 | Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000) |
| 8 | Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5) |
| 12 | Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5) |

DIMZIN also affects real-to-string conversions performed by the AutoLISP `rtos` and `angtos` functions.

# DISPSILH

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**0
Controls display of silhouette edges of 3D solid objects in a 2D wireframe or 3D wireframe visual style.

You can use the REGEN command to display the results.

DISPSILH also suppressess the mesh displayed when using the HIDE command in the 2D wireframe visual style.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# DISTANCE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Not-saved
**Initial value:**None
Stores the distance computed by the DIST command.

# DONUTID

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:**0.5000
Sets the default for the inside diameter of a donut.

# DONUTOD

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:** 1.0000
Sets the default for the outside diameter of a donut. The value must be nonzero. If DONUTID is larger than DONUTOD, the two values are swapped by the next command.

# DRAGMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2
Controls the display of objects being dragged. This system variable has the same name as a command. Use the SETVAR command to access this system variable.

When it is on, the image of an object is displayed as you drag it to another position. With some computer configurations, dragging can be time-consuming. Use DRAGMODE to suppress dragging.

| | |
|---|---|
| 0 | Does not display an outline of the object as you drag it |
| 1 | Displays the outline of the object as you drag it only if you enter **drag** at the command prompt after selecting the object to drag |
| 2 | Auto; always displays an outline of the object as you drag it |

# DRAGP1

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 10
Sets the regen-drag input sampling rate.

# DRAGP2

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 25
Sets the fast-drag input sampling rate.

# DRAGVS

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** Current visual style
Sets the visual style while creating 3D solid primitives and extruded solids and surfaces. You can enter a period (.) to specify the current visual style. DRAGVS can only be set to a visual style that is saved in the drawing.

DRAGVS has no effect when the current viewport's visual style is set to 2D Wireframe.

The visual style specified for DRAGVS cannot be removed with the *PURGE* command.

# DRAWORDERCTL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 3

Controls the display order of overlapping objects. Use this setting to improve the speed of editing operations in large drawings. The commands that are affected by inheritance are BREAK, FILLET, HATCH, HATCHEDIT, EXPLODE, TRIM, JOIN, PEDIT, and OFFSET.

| | |
|---|---|
| 0 | Turns off the default draw order of overlapping objects: after objects are edited, regardless of their draw order, the objects are displayed on top until a drawing is regenerated (REGEN) or reopened. This setting also turns off draw order inheritance: new objects that are created from another object using the commands listed above are not assigned the draw order of the original object. |
| 1 | Turns on the default draw order of objects: after objects are edited, they are automatically displayed according to the correct draw order. |
| 2 | Turns on draw order inheritance: new objects created from another object using the commands listed above are assigned the draw order of the original object. |
| 3 | Provides full draw order display. Turns on the correct draw order of objects, and turns on draw order inheritance. |

**NOTE** Full draw order display may slow some editing operations.

# DRSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** Varies
Determines whether the Drawing Recovery Manager window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# DTEXTED

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2
Specifies the user interface displayed for editing single-line text.

| | |
|---|---|
| 0 | Displays the In-Place Text Editor. |
| 1 | Displays a text prompt for creating text and allows you to click anywhere in the drawing to create a new text block; displays the Edit Text dialog box for editing text. |
| 2 | Displays the In-Place Text Editor for creating and editing text. When creating text, you can click anywhere in a drawing to create a new text block. You can also use the keyboard to move among text blocks (for example: for new text created using the *TEXT* command, you can navigate through text groups by pressing TAB or Shift+TAB, or edit a group of text lines by pressing ALT and clicking each text object.) |

# DWFFRAME

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Determines whether the DWF underlay frame is visible.

| | |
|---|---|
| 0 | The DWF underlay frame is not visible and it is not plotted |
| 1 | Displays the DWF underlay frame and allows it to be plotted |
| 2 | Displays the DWF underlay frame but keeps it from being plotted |

# DWFOSNAP

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Determines whether object snapping is active for geometry in DWF underlays that are attached to the drawing.

| | |
|---|---|
| 0 | Object snapping is disabled for geometry in all DWF underlay attachments in the drawing |
| 1 | Object snapping is enabled for geometry in all DWF underlay attachments in the drawing |

# DWGCHECK

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Checks drawings for potential problems when opening them.

| | |
|---|---|
| 0 | If a drawing that you try to open has a potential problem, you are warned before the drawing is opened. |
| 1 | If a drawing that you try to open has a potential problem, or if it was saved by an application other than one released by Autodesk or based on RealDWG, you are warned before the drawing is opened. |
| 2 | If a drawing that you try to open has a potential problem, you are notified at the command prompt. |
| 3 | If a drawing that you try to open has a potential problem, you are notified at the command prompt. If the drawing has a potential problem and the drawing was saved by an application other than one released by Autodesk or based on RealDWG, you are warned before the drawing is opened. |

# DWGCODEPAGE

## Quick Reference

(Read-only)
**Type:** String
**Saved in:**Drawing
Stores the same value as SYSCODEPAGE (for compatibility reasons).

# DWGNAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**Drawing1.dwg
Stores the name of the current drawing.

If a new drawing has not been saved yet, DWGNAME defaults to *Drawing1.dwg*. Additional new drawings are named *Drawing2.dwg*, *Drawing3.dwg*, and so on.

If you specified a drive and folder prefix, the prefix is stored in DWGPREFIX.

# DWGPREFIX

## Quick Reference

(Read-only)
**Type:** String
**Saved in:**Not-saved
Stores the drive and folder prefix for the drawing.

# DWGTITLED

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Indicates whether the current drawing has been named.

| | |
|---|---|
| 0 | Drawing has not been named |
| 1 | Drawing has been named |

# DXEVAL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 12
Controls when data extraction tables are compared against the data source, and if the data is not current, displays an update notification

The setting is stored as an integer using the sum of the following values:

| | |
|---|---|
| 0 | No notification |
| 1 | Open |
| 2 | Save |
| 4 | Plot |
| 8 | Publish |
| 16 | eTransmit/Archive |
| 32 | Save with Automatic Update |
| 64 | Plot with Automatic Update |
| 128 | Publish with Automatic Update |
| 256 | eTransmit/Archive with Automatic Update |

# DYNDIGRIP

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:** 31
Controls which dynamic dimensions are displayed during grip stretch editing. The DYNDIVIS system variable must be set to 2, which displays all dynamic dimensions.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | None |
| 1 | Resulting dimension |
| 2 | Length change dimension |
| 4 | Absolute angle dimension |
| 8 | Angle change dimension |
| 16 | Arc radius dimension |

The *DYNMODE* system variable turns Dynamic Input features on and off.

# DYNDIVIS

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 1

Controls how many dynamic dimensions are displayed during grip stretch editing. DYNDIGRIP controls which dynamic dimensions are displayed during grip stretch editing.

| | |
|---|---|
| 0 | Only the first dynamic dimension in the cycle order |
| 1 | Only the first two dynamic dimensions in the cycle order |
| 2 | All dynamic dimensions, as controlled by the *DYNDIGRIP* system variable |

The *DYNMODE* system variable turns Dynamic Input features on and off.

# DYNMODE

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 3
Turns Dynamic Input features on and off. When all features are on, the context governs what is displayed.

When DYNMODE is set to a negative value, the Dynamic Input features are not turned on, but the setting is stored. Press the Dyn button in the status bar to set DYNMODE to the corresponding positive value.

| | |
|---|---|
| 0 | All Dynamic Input features, including dynamic prompts, off |
| 1 | Pointer input on |
| 2 | Dimensional input on |
| 3 | Both pointer input and dimensional input on |

If dynamic prompts are on (*DYNPROMPT* is set to 1), they are displayed when DYNMODE is set to 1, 2, or 3.

When dimensional input is turned on (DYNMODE = 2 or 3), the program switches to pointer input when you enter a comma or an angle bracket (<), or when you select multiple grip points.

When DYNMODE is set to 1, 2, or 3, you can turn off all features temporarily by holding down the temporary override key, F12.

Settings are on the Dynamic Input tab in the Drafting Settings dialog box (page 488).

# DYNPICOORDS

## Quick Reference

**Type:** Switch
**Saved in:** User-settings
**Initial value:**0
Controls whether pointer input uses relative or absolute format for coordinates.

| | |
|---|---|
| 0 | Relative |
| 1 | Absolute |

The *DYNMODE* system variable turns Dynamic Input features on and off.

# DYNPIFORMAT

## Quick Reference

**Type:** Switch
**Saved in:** User-settings
**Initial value:**0
Controls whether pointer input uses polar or Cartesian format for coordinates. This setting applies only to a second or next point.

| | |
|---|---|
| 0 | Polar |
| 1 | Cartesian |

The *DYNMODE* system variable turns Dynamic Input features on and off.

# DYNPIVIS

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 1
Controls when pointer input is displayed.

| | |
|---|---|
| 0 | Only when you type at a prompt for a point |
| 1 | Automatically at a prompt for a point |
| 2 | Always |

The *DYNMODE* system variable turns Dynamic Input features on and off.

# DYNPROMPT

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 1
Controls display of prompts in Dynamic Input tooltips.

| | |
|---|---|
| 0 | Off |
| 1 | On |

The *DYNMODE* system variable turns Dynamic Input features on and off. When DYNMODE is set to 0 (off), prompts are not displayed regardless of the DYNPROMPT setting.

# DYNTOOLTIPS

### Quick Reference

**Type:** Switch
**Saved in:** User-settings
**Initial value:** 1
Controls which tooltips are affected by tooltip appearance settings.

| | |
|---|---|
| 0 | Only Dynamic Input value fields |
| 1 | All drafting tooltips |

# E System Variables

# EDGEMODE

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

| | |
|---|---|
| 0 | Uses the selected edge without an extensions |
| 1 | Extends or trims the selected object to an imaginary extension of the cutting or boundary edge |

Lines, arcs, elliptical arcs, rays, and polylines are objects eligible for natural extension. The natural extension of a line or ray is an unbounded line (xline), an arc is a circle, and an elliptical arc is an ellipse. A polyline is broken down into its line and arc components, which are extended to their natural boundaries.

# ELEVATION

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0000
Stores the current elevation of new objects relative to the current UCS.

# ENTERPRISEMENU

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"."
Displays the file name for the enterprise CUI (if defined), including the path for the file name.

# ERRNO

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Displays the number of the appropriate error code when an AutoLISP function call causes an error that AutoCAD detects. AutoLISP applications can inspect the current value of ERRNO with (getvar "errno").

The ERRNO system variable is not always cleared to zero. Unless it is inspected immediately after an AutoLISP function has reported an error, the error that its value indicates may be misleading. This variable is always cleared when starting or opening a drawing.

See the *AutoLISP Developer's Guide* for more information.

# ERSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** Varies
Determines whether the External References palette is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# EXPERT

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Controls whether certain prompts are issued.

| | |
|---|---|
| 0 | Issues all prompts normally. |
| 1 | Suppresses "About to regen, proceed?" and "Really want to turn the current layer off?" (*-LAYER*) |
| 2 | Suppresses the preceding prompts and "Block already defined. Redefine it?" (*-BLOCK*) and "A drawing with this name already exists. Overwrite it?" (*SAVE* or *WBLOCK*). |
| 3 | Suppresses the preceding prompts and those issued by the *LINETYPE* command line (-LINETYPE) if you try to load a linetype that's already loaded or create a new linetype in a file that already defines that linetype. |

| | |
|---|---|
| 4 | Suppresses the preceding prompts and those issued by *UCS* Save and *VPORTS* Save if the name you supply already exists. |
| 5 | Suppresses the prompt, "That name is already in Use, re-define it?" issued by the -*DIMSTYLE* Save option when you supply the name of an existing dimension style. Suppresses the same prompt issued by the -SCALELISTEDIT (page 1247) Add option. |

When a prompt is suppressed by EXPERT, the operation in question is performed as though you entered **y** at the prompt. Setting EXPERT can affect scripts, menu macros, AutoLISP, and the command functions.

# EXPLMODE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 1
Controls whether the EXPLODE command supports nonuniformly scaled (NUS) blocks.

| | |
|---|---|
| 0 | Does not explode NUS blocks |
| 1 | Explodes NUS blocks |

# EXTMAX

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** Varies

Stores the upper-right point of the drawing extents. Expands outward as new objects are drawn; shrinks only with ZOOM All or ZOOM Extents. Reported in world coordinates for the current space.

# EXTMIN

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** Varies
Stores the lower-left point of the drawing extents. Expands outward as new objects are drawn; shrinks only with ZOOM All or ZOOM Extents. Reported in world coordinates for the current space.

# EXTNAMES

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

| | |
|---|---|
| 0 | Uses AutoCAD Release 14 parameters, which limit names to 31 characters in length. Names can include the letters A to Z, the numerals 0 to 9, and the special characters dollar sign ($), underscore (_), and hyphen (-). |
| 1 | Uses AutoCAD 2000 (and later releases) parameters. Names can be up to 255 characters in length, and can include the letters A to Z, the numerals 0 to 9, spaces, and any special characters not used by Microsoft® Windows® and AutoCAD for other purposes. |

# F System Variables

## FACETRATIO

### Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls the aspect ratio of faceting for cylindrical and conic solids. A setting of 1 increases the density of the mesh to improve the quality of rendered and shaded models.

| | |
|---|---|
| 0 | Creates an *N* by 1 mesh for cylindrical and conic solids |
| 1 | Creates an *N* by *M* mesh for cylindrical and conic solids |

## FACETRES

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.5
Adjusts the smoothness of shaded objects and objects with hidden lines removed. Valid values are from 0.01 to 10.0.

**NOTE** 3DDWFPREC (page 1603) replaces FACETRES in controlling the precision of published 3D DWF files.

# FIELDDISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether fields are displayed with a gray background. The background is not plotted.

| | |
|---|---|
| 0 | Fields are displayed with no background |
| 1 | Fields are displayed with a gray background |

# FIELDEVAL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**31
Controls how fields are updated. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Not updated |
| 1 | Updated on open |
| 2 | Updated on save |
| 4 | Updated on plot |
| 8 | Updated on use of ETRANSMIT |
| 16 | Updated on regeneration |

**NOTE** The Date field is updated by *UPDATEFIELD*, but it is not updated automatically based on the setting of the FIELDEVAL system variable.

# FILEDIA

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Suppresses display of file navigation dialog boxes.

| | |
|---|---|
| 0 | Does not display dialog boxes. You can still request a file dialog box to appear by entering a tilde (~) in response to the command's prompt. The same is true for AutoLISP and ADS functions. |
| 1 | Displays dialog boxes. However, if a script or AutoLISP/ObjectARX™ program is active, an ordinary prompt is displayed. |

# FILLETRAD

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Stores the current fillet radius.

# FILLMODE

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**1
Specifies whether hatches and fills, two-dimensional solids, and wide polylines are filled in.

| | |
|---|---|
| 0 | Objects are not filled |
| 1 | Objects are filled |

# FONTALT

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**simplex.shx
Specifies the alternate font to be used when the specified font file cannot be located. When a drawing file with a defined text style is opened and an alternate font is not specified, the Alternate Font dialog box is displayed.

# FONTMAP

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**acad.fmp
Specifies the font mapping file to be used. A font mapping file contains one font mapping per line; the original font used in the drawing and the font to be substituted for it are separated by a semicolon (;). For example, to substitute the Times TrueType font for the Roman font, the line in the mapping file would read as follows:

```
romanc.shx;times.ttf
```

If FONTMAP does not point to a font mapping file, if the FMP file is not found, or if the font file name specified in the FMP file is not found, the font defined

in the style is used. If the font in the style is not found, a font is substituted according to the substitution rules.

FONTMAP only works with text created through the *MTEXT* command.

# FRONTZ

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** None
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units. Meaningful only if the *front clipping* and *front clip not at eye* bitcodes in *VIEWMODE* are on. The FRONTZ value is the last front clipping plane value set current with the CAMERA, DVIEW, or 3DCLIP command. The distance of the front clipping plane from the camera point is found by subtracting FRONTZ from the camera-to-target distance.

# FULLOPEN

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
Indicates whether the current drawing is partially open.

| | |
|---|---|
| 0 | Indicates a partially open drawing |
| 1 | Indicates a fully open drawing |

# FULLPLOTPATH

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether the full path of the drawing file is sent to the plot spooler.

| | |
|---|---|
| 0 | Sends the drawing file name only |
| 1 | Sends the full path of the drawing file |

# G System Variables

# GRIDDISPLAY

### Quick Reference

**Type:** Bitcode
**Saved in:** Drawing
**Initial value:** 2
Controls the display behavior and display limits of the grid.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Restricts the grid to the area specified by the LIMITS command |
| 1 | Does not restrict the grid to the area specified by the LIMITS command |
| 2 | Turns on adaptive grid display, which limits the density of the grid when zoomed out |

| | |
|---|---|
| 4 | If the grid is set to adaptive display and when zoomed in, generates additional, more closely spaced grid lines in the same proportion as the intervals of the major grid lines |
| 8 | Changes the grid plane to follow the XY plane of the dynamic UCS. |

**NOTE** Setting 4 is ignored unless setting 2 is specified.

# GRIDMAJOR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**5
Controls the frequency of major grid lines compared to minor grid lines. Valid values range from 1 to 100.

Grid lines are displayed in any visual style except 2D Wireframe.

# GRIDMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Specifies whether the grid is turned on or off.

| | |
|---|---|
| 0 | Turns the grid off |
| 1 | Turns the grid on |

# GRIDUNIT

## Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:** 0.5000,0.5000 (imperial) or 10,10 (metric)
Specifies the grid spacing (X and Y) for the current viewport.

# GRIPBLOCK

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls the display of grips on nested objects in blocks when selected.

| | |
|---|---|
| 0 | Assigns a grip only to the insertion point of the block |
| 1 | Assigns grips to objects within the block |

# GRIPCOLOR

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 160
Controls the color of nonselected grips. The valid range is 1 to 255.

# GRIPDYNCOLOR

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 140
Controls the color of custom grips for dynamic blocks. The valid range is 1 to 255.

# GRIPHOT

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the color of selected grips. The valid range is 1 to 255.

# GRIPHOVER

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 3
Controls the fill color of an unselected grip when the cursor pauses over it. The valid range is 1 to 255.

# GRIPOBJLIMIT

### Quick Reference

**Type:** Integer
**Saved in:** Registry

**Initial value:**100
Suppresses the display of grips when the selection set includes more than the specified number of objects. The valid range is 0 to 32,767. For example, when set to 1, grips are suppressed when more than one object is selected. When set to 0, grips are always displayed.

# GRIPS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

| | |
|---|---|
| 0 | Turns off grips |
| 1 | Turns on grips |

To adjust the size of the grips and the effective selection area used by the cursor when you snap to a grip, use *GRIPSIZE*.

# GRIPSIZE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**5
Sets the size of the grip box in pixels. The valid range is 1 to 255.

# GRIPTIPS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

| | |
|---|---|
| 0 | Turns off the display of grip tips |
| 1 | Turns on the display of grip tips |

# GTAUTO

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether or not grip tools display automatically when selecting objects before starting a command in a viewport set to a 3D visual style.

| | |
|---|---|
| 0 | Grip tools do not display automatically when selecting objects before starting a command. |
| 1 | Grip tools display automatically after creating a selection set before starting a command. |

# GTDEFAULT

## Quick Reference

**Type:** Integer

**Saved in:** Registry
**Initial value:**0
Controls whether or not the 3DMOVE and 3DROTATE commands start automatically when the MOVE and ROTATE commands (respectively) are started in a 3D view.

| | |
|---|---|
| 0 | *3DMOVE* and *3DROTATE* commands do not start automatically when the *MOVE* and *ROTATE* commands (respectively) are started in a 3D view. |
| 1 | *3DMOVE* and *3DROTATE* commands start automatically when the *MOVE* and *ROTATE* commands (respectively) are started in a 3D view. |

# GTLOCATION

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls the initial location of grip tools when objects are selected prior to running the 3DMOVE or 3DROTATE commands.

| | |
|---|---|
| 0 | Places the grip tool at the same location as the UCS icon. Also aligns the grip tool with the UCS icon. |
| 1 | Places the grip tool on and aligned with the last selected object or subobject. |

# H System Variables

## HALOGAP

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Specifies a gap to be displayed where an object is hidden by another object. The value is specified as a percent of one unit and is independent of the zoom level.

HALOGAP is available only in 2D views. In 3D views, the *VSHALOGAP* system variable is used.

## HANDLES

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Drawing
**Initial value:**On
Reports whether object handles can be accessed by applications. Because handles can no longer be turned off, has no effect except to preserve the integrity of scripts.

## HIDEPRECISION

### Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0

Controls the accuracy of hides and shades. Hides can be calculated in double precision or single precision. Setting HIDEPRECISION to 1 produces more accurate hides by using double precision, but this setting also uses more memory and can affect performance, especially when hiding solids.

| | |
|---|---|
| 0 | Single precision; uses less memory |
| 1 | Double precision; uses more memory |

# HIDETEXT

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** On
Specifies whether text objects created by the TEXT, DTEXT, or MTEXT command are processed during a HIDE command.

| | |
|---|---|
| Off | Text is not hidden and does not hide other objects |
| On | Text is hidden but does not hide other objects |

# HIGHLIGHT

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 1
Controls object highlighting; does not affect objects selected with grips.

| | |
|---|---|
| 0 | Turns off object selection highlighting |
| 1 | Turns on object selection highlighting |

# HPANG

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:**0
Specifies the hatch pattern angle.

# HPASSOC

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether hatch patterns and gradient fills are associative.

| | |
|---|---|
| 0 | Hatch patterns and gradient fills are not associated with their boundaries |
| 1 | Hatch patterns and gradient fills are associated with their boundaries and are updated when the boundaries change |

# HPBOUND

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Controls the object type created by the BHATCH and BOUNDARY commands.

| | |
|---|---|
| 0 | Creates a region |

| | |
|---|---|
| 1 | Creates a polyline |

# HPDOUBLE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Specifies hatch pattern doubling for user-defined patterns. Doubling specifies a second set of lines drawn at 90 degrees to the original lines.

| | |
|---|---|
| 0 | Turns off hatch pattern doubling |
| 1 | Turns on hatch pattern doubling |

# HPDRAWORDER

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 3
Controls the draw order of hatches and fills. Stores the Draw Order setting from the Hatch and Fill Dialog Box.

| | |
|---|---|
| 0 | None. The hatch or fill is not assigned a draw order. |
| 1 | Send to back. The hatch or fill is sent to the back of all other objects. |
| 2 | Bring to front. The hatch or fill is brought to the front of all other objects. |

| | |
|---|---|
| 3 | Send behind boundary. The hatch or fill is sent behind the hatch boundary. |
| 4 | Bring in front of boundary. The hatch or fill is brought in front of the hatch boundary |

# HPGAPTOL

## Quick Reference

**Type:** Real
**Saved in:** Registry
**Initial value:** 0
Treats a set of objects that almost enclose an area as a closed hatch boundary. The default value, 0, specifies that the objects enclose the area, with no gaps. Enter a value, in drawing units, from 0 to 5000 to set the maximum size of gaps that can be ignored when the objects serve as a hatch boundary.

# HPMAXLINES

## Quick Reference

**Type:** Real
**Saved in:** Registry
**Initial value:** 1000000
Controls the maximum number of hatch lines that will generate. Values can be set at a minimum of 100 and a maximum of 10,000,000.

# HPINHERIT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0

Controls the hatch origin of the resulting hatch when using Inherit Properties in HATCH and HATCHEDIT.

| | |
|---|---|
| 0 | The hatch origin is taken from HPORIGIN |
| 1 | The hatch origin is taken from the source hatch object |

# HPNAME

## Quick Reference

**Type:** String
**Saved in:** Not-saved
**Initial value:** ANSI31
Sets a default hatch pattern name of up to 34 characters without spaces. Returns "" if there is no default. Enter a period (.) to reset HPNAME to the default value.

# HPOBJWARNING

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 10000
Sets the number of hatch boundary objects that can be selected before displaying a warning message. The maximum value can vary, but is significantly larger than 100000000 (one hundred million).

# HPORIGIN

## Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:** 0,0

Sets the hatch origin point for new hatch objects relative to the current user coordinate system.

# HPORIGINMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls how HATCH determines the default hatch origin point.

| | |
|---|---|
| 0 | Hatch origins are set using HPORIGIN |
| 1 | Hatch origins are set using the bottom-left corner of the rectangular extents of the hatch boundaries |
| 2 | Hatch origins are set using the bottom-right corner of the rectangular extents of the hatch boundaries |
| 3 | Hatch origins are set using the top-right corner of the rectangular extents of the hatch boundaries |
| 4 | Hatch origins are set using the top-left corner of the rectangular extents of the hatch boundaries |
| 5 | Hatch origins are set using the center of the rectangular extents of the hatch boundaries |

# HPSCALE

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:** 1.0000

Specifies the hatch pattern scale factor, which must be greater than zero.

# HPSEPARATE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether HATCH creates a single hatch object or separate hatch objects when operating on several closed boundaries.

| | |
|---|---|
| 0 | A single hatch object is created |
| 1 | Separate hatch objects are created |

# HPSPACE

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:**1.0000
Specifies the hatch pattern line spacing for user-defined simple patterns, which must be greater than zero.

# HYPERLINKBASE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**""
Specifies the path used for all relative hyperlinks in the drawing. If no value is specified, the drawing path is used for all relative hyperlinks.

# I System Variables

## IMAGEHLT

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether the entire raster image or only the raster image frame is highlighted.

| | |
|---|---|
| 0 | Highlights only the raster image frame |
| 1 | Highlights the entire raster image |

## IMPLIEDFACE

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the detection of implied faces. This variable must be set to 1 if you want to select and modify implied faces.

| | |
|---|---|
| 0 | Implied faces cannot be detected. |
| 1 | Implied faces can be detected. |

# INDEXCTL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls whether layer and spatial indexes are created and saved in drawing files.

| | |
|---|---|
| 0 | No indexes are created |
| 1 | Layer index is created |
| 2 | Spatial index is created |
| 3 | Layer and spatial indexes are created |

# INETLOCATION

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** http://www.autodesk.com
Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.

# INPUTHISTORYMODE

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:** 15

Controls the content and location of the display of a history of user input.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | No history of recent input is displayed. |
| 1 | History of recent input is displayed at the command line or in a dynamic prompt tooltip. Access with the Up Arrow and Down Arrow keys at the command prompt, or at an input prompt. |
| 2 | History of recent input for the current *command* is displayed in the shortcut menu under Recent Input. |
| 4 | History of recent input for all *commands* in the current session is displayed in the shortcut menu under Recent Input. |
| 8 | Markers for recent input of point locations are displayed. Use the arrow keys on the command line at a prompt for specifying a point location. |

# INSBASE

## Quick Reference

**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** 0.0000,0.0000,0.0000
Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.

# INSNAME

## Quick Reference

**Type:** String

**Saved in:** Not-saved
**Initial value:**""
Sets a default block name for the INSERT command. The name must conform to symbol naming conventions. Returns "" if no default is set. Enter a period (.) to set no default.

# INSUNITS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs inserted or attached to a drawing.

**NOTE** The INSUNITS setting is ignored when inserting blocks into a drawing.

| | |
|---|---|
| 0 | Unspecified (No units) |
| 1 | Inches |
| 2 | Feet |
| 3 | Miles |
| 4 | Millimeters |
| 5 | Centimeters |
| 6 | Meters |
| 7 | Kilometers |
| 8 | Microinches |
| 9 | Mils |

| | |
|---|---|
| 10 | Yards |
| 11 | Angstroms |
| 12 | Nanometers |
| 13 | Microns |
| 14 | Decimeters |
| 15 | Dekameters |
| 16 | Hectometers |
| 17 | Gigameters |
| 18 | Astronomical Units |
| 19 | Light Years |
| 20 | Parsecs |

# INSUNITSDEFSOURCE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Sets source content units value when INSUNITS is set to 0. Valid range is 0 to 20.

| | |
|---|---|
| 0 | Unspecified (No units) |
| 1 | Inches |

| | |
|---|---|
| 2 | Feet |
| 3 | Miles |
| 4 | Millimeters |
| 5 | Centimeters |
| 6 | Meters |
| 7 | Kilometers |
| 8 | Microinches |
| 9 | Mils |
| 10 | Yards |
| 11 | Angstroms |
| 12 | Nanometers |
| 13 | Microns |
| 14 | Decimeters |
| 15 | Dekameters |
| 16 | Hectometers |
| 17 | Gigameters |
| 18 | Astronomical Units |
| 19 | Light Years |

| | |
|---|---|
| 20 | Parsecs |

# INSUNITSDEFTARGET

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Sets target drawing units value when INSUNITS is set to 0. Valid range is 0 to 20.

| | |
|---|---|
| 0 | Unspecified (No units) |
| 1 | Inches |
| 2 | Feet |
| 3 | Miles |
| 4 | Millimeters |
| 5 | Centimeters |
| 6 | Meters |
| 7 | Kilometers |
| 8 | Microinches |
| 9 | Mils |
| 10 | Yards |
| 11 | Angstroms |

| | |
|---|---|
| 12 | Nanometers |
| 13 | Microns |
| 14 | Decimeters |
| 15 | Dekameters |
| 16 | Hectometers |
| 17 | Gigameters |
| 18 | Astronomical Units |
| 19 | Light Years |
| 20 | Parsecs |

# INTELLIGENTUPDATE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**20
Controls the graphics refresh rate. The default value is 20 frames per second. If you encounter problems related to graphics generation or timing, turn off the variable by setting it to 0. INTELLIGENTUPDATE works by suppressing the graphics update until the timer expires. Subsequent updates reset the timer.

The performance improvement significantly affects updates for scripts and AutoLISP graphics. Those using regular AutoCAD commands will not see a noticeable difference in performance.

# INTERFERECOLOR

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**1
Sets the color for interference objects. Valid values include BYLAYER, BYBLOCK, a color name, and integers from 0 to 255.

Valid values for True Colors are a string of integers each from 0 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

If you have a color book installed, you can specify any colors that are defined in the book.

# INTERFEREOBJVS

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**Realistic
Sets the visual style for interference objects. INTERFEREOBJVS can only be set to a visual style that is saved in the drawing.

The visual style specified for INTERFEREOBJVS cannot be removed with the *PURGE* command.

# INTERFEREVPVS

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**3D Wireframe

Specifies the visual style for the viewport during interference checking. INTERFEREVPVS can only be set to a visual style that is saved in the drawing.

The visual style specified for INTERFEREVPVS cannot be removed with the *PURGE* command.

# INTERSECTIONCOLOR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**257
Controls the color of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.

Value 0 designates ByBlock, value 256 designates ByLayer, and value 257 designates ByEntity. Values 1-255 designate an AutoCAD Color Index (ACI) color. True Colors and Color Book colors are also available.

The INTERSECTIONCOLOR setting is visible only if *INTERSECTIONDISPLAY* is turned on.

# INTERSECTIONDISPLAY

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:**Off
Controls the display of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.

| | |
|---|---|
| Off | Turns off the display of intersection polylines |
| On | Turns on the display of intersection polylines |

# ISAVEBAK

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Improves the speed of incremental saves, especially for large drawings. ISAVEBAK controls the creation of a backup file (BAK). In the operating system, copying the file data to create a BAK file for large drawings takes a major portion of the incremental save time.

| | |
|---|---|
| 0 | No BAK file is created (even for a full save) |
| 1 | A BAK file is created |

**WARNING** In some cases (such as a power failure in the middle of a save), it's possible that drawing data can be lost.

# ISAVEPERCENT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**50
Determines the amount of wasted space tolerated in a drawing file. The value of ISAVEPERCENT is an integer between 0 and 100. The default value of 50 means that the estimate of wasted space within the file does not exceed 50 percent of the total file size. Wasted space is eliminated by periodic full saves. When the estimate exceeds 50 percent, the next save will be a full save. This resets the wasted space estimate to 0. If ISAVEPERCENT is set to 0, every save is a full save.

## ISOLINES

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**4
Specifies the number of contour lines per surface on objects. Valid integer values are from 0 to 2047.

Valid settings are integers from 0 to 2047.

# L System Variables

## LASTANGLE

### Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Not-saved
**Initial value:**0
Stores the end angle of the last arc entered relative to the XY plane of the current UCS for the current space.

## LASTPOINT

### Quick Reference

**Type:** 3D-point
**Saved in:** Not-saved
**Initial value:**0.0000,0.0000,0.0000
Stores the last point entered, expressed as a UCS coordinate for the current space; referenced by the at symbol (@) during keyboard entry.

# LASTPROMPT

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Stores the last string echoed to the command line. This string is identical to the last line seen at the command line and includes any user input.

# LATITUDE

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**Varies
Specifies the latitude of the drawing model in decimal format. The default is the latitude of San Francisco, California. The valid range is -90 to +90. Positive values represent north latitudes. This value is not affected by the settings of the AUNITS and AUPREC system variables.

# LAYEREVAL

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls when the Unreconciled New Layer filter list in the Layer Properties Manager is evaluated for new layers.

The setting is stored in an integer using one of the following values:

| | |
|---|---|
| 0 | Off |

| | |
|---|---|
| 1 | Detects when new xref layers have been added in the drawing |
| 2 | Detects when new layers have been added in the drawing and xrefs |

# LAYERFILTERALERT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Deletes excessive layer filters to improve performance. When a drawing has 100 or more layer filters, and the number of layer filters exceeds the number of layers, LAYERFILTERALERT provides a method for deleting layer filters to improve performance.

| | |
|---|---|
| 0 | Off |
| 1 | When the Layer Manager is opened, deletes all layer filters; no message is displayed |
| 2 | When the Layer Manager is opened, displays a message that states the problem, recommends deleting all filters, and offers a choice: "Do you want to delete all layer filters now?" |
| 3 | When the drawing is opened, displays a message that states the problem and offers to display a dialog box where you can choose which filters to delete |

# LAYERNOTIFY

## Quick Reference

**Type:** Bitcode
**Saved in:** Drawing
**Initial value:**15
Specifies when an alert displays for new layers that have not yet been reconciled.

| | |
|---|---|
| 0 | Off |
| 1 | Plot |
| 2 | Open |
| 4 | Load/Reload/Attach for xrefs |
| 8 | Restore layer state |
| 16 | Save |
| 32 | Insert |

# LAYLOCKFADECTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**50
Controls the dimming for objects on locked layers

The range for controlling the dimming for objects on locked layers is from -90 to 90.

| | |
|---|---|
| 0 | Locked layers are not dimmed. |
| >0 | When the value is positive, controls the percent dimming up to 90 percent. |
| <0 | When the value is negative, locked layers are not dimmed, but the value is saved for switching to that value by changing the sign. |

**NOTE** The dimming value is limited to 90 percent to avoid confusion with layers that are turned off or frozen.

# LAYOUTREGENCTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Specifies how the display list is updated in the Model tab and layout tabs. For each tab, the display list is updated either by regenerating the drawing when you switch to that tab or by saving the display list to memory and regenerating only the modified objects when you switch to that tab. Changing the LAYOUTREGENCTL setting can improve performance.

| | |
|---|---|
| 0 | The drawing is regenerated each time you switch tabs. |
| 1 | For the Model tab and the last layout made current, the display list is saved to memory and regenerations are suppressed when you switch between the two tabs. For all other layouts, regenerations still occur when you switch to those tabs. |
| 2 | The drawing is regenerated the first time you switch to each tab. For the remainder of the drawing session, the |

| | |
|---|---|
| | display list is saved to memory and regenerations are suppressed when you switch to those tabs. |

The performance gain achieved by changing the LAYOUTREGENCTL setting is dependent on several factors, including the drawing size and type, the objects contained in the drawing, the amount of available memory, and the effect of other open drawings or applications. When LAYOUTREGENCTL is set to 1 or 2, the amount of additional memory used is the size of the Model tab's display list multiplied by the number of viewports in each layout for which the display list is saved.

If LAYOUTREGENCTL is set to 1 or 2 and performance seems slow in general or when you switch between tabs for which the display list is saved, consider changing to a setting of 0 or 1 to find the optimal balance for your work environment. For additional information about performance settings, see System tab (in the Options dialog box).

Regardless of the LAYOUTREGENCTL setting, if you redefine a block or undo a tab switch, the drawing is regenerated the first time you switch to any tab that contains saved viewports.

# LEGACYCTRLPICK

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0

Specifies the keys for selection cycling and the behavior for CTRL + left-click.

Specifies the keys for selection cycling and the behavior for CTRL + left-click.

| | |
|---|---|
| 0 | CTRL + left-click is used to select subobjects (faces, edges, and vertices) on 3D solids. |
| 1 | CTRL + left-click is used to cycle through overlapping objects. Disallows using CTRL + left-click to select subobjects on 3D solids. |

# LENSLENGTH

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 50.0000
Stores the length of the lens (in millimeters) used in perspective viewing.

# LIGHTGLYPHDISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls whether light glyphs are displayed. When this system variable is set to Off, the glyphs that represent lights in the drawing are not displayed.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# LIGHTINGUNITS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls whether generic or photometric lights are used, and indicates the current lighting units

When this system variable is set to 1 or 2, photometric lighting is enabled; otherwise standard (generic) lighting is used.

| | |
|---|---|
| 0 | No lighting units are used and standard (generic) lighting is enabled |
| 1 | International lighting units are used and photometric lighting is enabled |
| 2 | American lighting units are used and photometric lighting is enabled |

# LIGHTLISTSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Indicates whether the Lights in Model window is open.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# LIGHTSINBLOCKS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls whether lights contained in blocks are used when rendering

By default, this system variable is turned on in AutoCAD 2008 drawings. When this system variable is off, lights in blocks do not affect 3D objects in the current viewport when rendering. Previous versions of AutoCAD did not support rendering lights in blocks.

| | |
|---|---|
| 0 (Off) | Lights in blocks are disabled |
| 1 (On) | Lights in blocks are enabled |

# LIMCHECK

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the creation of objects outside the grid limits.

| | |
|---|---|
| 0 | Objects can be created outside the limits |
| 1 | Objects cannot be created outside the limits |

# LIMMAX

## Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:**12.0000,9.0000

Stores the upper-right grid limits for the current space, expressed as a world coordinate. LIMMAX is read-only when paper space is active and the paper background or printable area is displayed.

# LIMMIN

### Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:** 0.0000,0.0000
Stores the lower-left grid limits for the current space, expressed as a world coordinate. LIMMIN is read-only when paper space is active and the paper background or printable area is displayed.

# LINEARBRIGHTNESS

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the global brightness level of the drawing in the standard lighting workflow.

Controls the global brightness level of the drawing when standard lighting is enabled (LIGHTINGUNITS system variable is set to 0). Valid settings are integers from -10 to 10. The global brutishness level in photometric lighting can be controlled by the LOGEXPBRIGHTNESS systems variable.

# LINEARCONTRAST

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the global contrast level of the drawing in the standard lighting workflow.

Controls the global contrast level of the drawing when standard lighting is enabled (LIGHTINGUNITS system variable is set to 0). Valid settings are integers

from -10 to 10. The global contrast level in photometric lighting can be controlled by LOGEXPCONTRAST.

# LOCALE

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Displays a code that indicates the current locale. This code appears as a three-letter abbreviation returned by the Windows GetLocaleInfo function using the LOCALE_SABBREVLANGNAME constant.

# LOCALROOTPREFIX

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"pathname"
Stores the full path to the root folder where local customizable files were installed. These files are stored in the product folder under the Local Settings folder; for example, "c:\Documents and Settings\username\Local Settings\Application Data\application_name\release_number\language".

The *Template* and *Textures* folders are in this location, and you can add any customizable files that you do not want to roam on the network. See ROAMABLEROOTPREFIX for the location of the roamable files.

# LOCKUI

## Quick Reference

**Type:** Bitcode

**Saved in:** Registry
**Initial value:**0
Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette. Locked toolbars and windows can still be opened and closed and items can be added and deleted. To unlock them temporarily, hold down CTRL.

A lock icon in the status bar tray indicates whether toolbars and windows are locked. Right-click the icon to display locking options.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Toolbars and windows not locked |
| 1 | Docked toolbars locked |
| 2 | Docked or anchored windows locked |
| 4 | Floating toolbars locked |
| 8 | Floating windows locked |

# LOFTANG1

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**90
Sets the draft angle through the first cross section in a loft operation. The 0 direction is measured outward from the curve on the plane of the curve. The positive direction is measured toward the next cross section. Valid values include 0 to less than 360.

# LOFTANG2

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**90
Sets the draft angle through the last cross section in a loft operation. The 0 direction is measured outward from the curve on the plane of the curve. The positive direction is measured toward the previous cross section. Valid values include 0 to less than 360.

# LOFTMAG1

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0000
Sets the magnitude of the draft angle through the first cross section in a loft operation. Controls how soon the surface starts bending back toward the next cross section.

# LOFTMAG2

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0000
Sets the magnitude of the draft angle through the last cross section in a loft operation. Controls how soon the surface starts bending back toward the next cross section.

# LOFTNORMALS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls the normals of a lofted object where it passes through cross sections. This setting is ignored when specifying a path or guide curves.

These settings can also be specified in the Loft Settings Dialog Box (page 765).

| | |
|---|---|
| 0 | Ruled |
| 1 | Smooth Fit |
| 2 | Start cross section |
| 3 | End cross section |
| 4 | Start and End cross sections |
| 5 | All cross sections |
| 6 | Use draft angle and magnitude |

# LOFTPARAM

## Quick Reference

**Type:** Bitcode
**Saved in:** Drawing
**Initial value:** 7
Controls the shape of lofted solids and surfaces.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 1 | No twist (minimizes the twist between cross sections) |
| 2 | Align direction (aligns the start to end direction of each cross section curve) |
| 4 | Simplify (produces simple solids and surfaces, such as a cylinder or plane, instead of spline solids and surfaces) |
| 8 | Close (closes the surface or solid between the first and the last cross sections) |

To specify more than one setting to control the shape of lofted solids or surfaces, enter the sum of their values. For example, entering 7 specifies the No twist (bitcode 1), Align direction (bitcode 2), and Simply (bitcode 4) settings.

# LOGEXPBRIGHTNESS

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**65.0
Controls the global brightness level of the drawing when using photometric lighting

Controls the global brightness level of the drawing when photometric lighting is enabled (LIGHTINGUNITS system variable is set to 1 or 2). Valid range is from 0.0 to 200.0. The global brightness level in standard lighting can be controlled by the LINEARBRIGHTNESS systems variable.

# LOGEXPCONTRAST

## Quick Reference

**Type:** Real
**Saved in:** Drawing

**Initial value:**50.0
Controls the global contrast level of the drawing when using photometric lighting

Controls the global contrast level of the drawing when photometric lighting is enabled (LIGHTINGUNITS system variable is set to 1 or 2). Valid range is from 0.0 to 100.0. The global contrast level in standard lighting can be controlled by the LINEARCONTRAST systems variable.

# LOGEXPDAYLIGHT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Controls if exterior daylight is used when using photometric lighting

When this system variable is off, exterior daylight is turned off when photometric lighting is enabled (LIGHTINGUNITS system variable is set to 1 or 2). When on, the exposure control algorithm in the rendering acts as if there is an extremely bright light source in the scene. Therefore, the default setting of "Auto" is tied directly to the sun setting. Under special circumstances, it may be desirable to manually set this variable due to extremely bright artificial light sources or, conversely, due to a sun-lit scene that it is illuminated only by a small amount of directly visible sunlight.

| | |
|---|---|
| 0 | Off |
| 1 | On |
| 2 | Auto, current sun status is used |

# LOGEXPMIDTONES

## Quick Reference

**Type:** Real

**Saved in:** Drawing
**Initial value:**1.0
Controls the global mid tones level of the drawing when using photometric lighting

Controls the global mid tones level of the drawing when photometric lighting is enabled (LIGHTINGUNITS system variable is set to 1 or 2). Valid range from 0.01 to 20.0.

# LOGFILEMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Specifies whether the contents of the text window are written to a log file.

| | |
|---|---|
| 0 | Log file is not maintained |
| 1 | Log file is maintained |

# LOGFILENAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Drawing
**Initial value:**Varies
Specifies the path and name of the text window log file for the current drawing. The initial value varies depending on the name of the current drawing and the installation folder.

# LOGFILEPATH

### Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**"<drive letter>:\Documents and Settings\<username>\Local Settings\Application Data\Autodesk\<application_name>\<release_number>\<locale_code>"
Specifies the path for the text window log files for all drawings in a session. You can also specify the path by using the OPTIONS command. The initial value is based on the installation folder.

# LOGINNAME

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Displays the user's name as configured or as input when the program starts. The maximum length for a login name is 30 characters.

# LONGITUDE

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**Varies
Specifies the longitude of the drawing model in decimal format. The default is the longitude of San Francisco, California. The valid range is -180 to +180. Positive values represent west longitudes. This value is not affected by the settings of the AUNITS and AUPREC system variables.

# LTSCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 1.0000
Sets the global linetype scale factor. The linetype scale factor cannot equal zero. This system variable has the same name as a command. Use the SETVAR command to access this system variable.

# LUNITS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 2
Sets linear units.

| | |
|---|---|
| 1 | Scientific |
| 2 | Decimal |
| 3 | Engineering |
| 4 | Architectural |
| 5 | Fractional |

# LUPREC

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**4
Sets the number of decimal places displayed for all read-only linear units, and for all editable linear units whose precision is less than or equal to the current LUPREC value. For editable linear units whose precision is greater than the current LUPREC value, the true precision is displayed. LUPREC does not affect the display precision of dimension text (see DIMSTYLE).

# LWDEFAULT

## Quick Reference

**Type:** Enum
**Saved in:** Registry
**Initial value:**25
Sets the value for the default lineweight. The default lineweight can be set to any valid lineweight value in hundredths of millimeters, including: 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.

All values must be entered in hundredths of millimeters. (Multiply a value by 2540 to convert values from inches to hundredths of millimeters.)

# LWDISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**OFF (0)
Controls whether the lineweight is displayed. The setting is saved with each tab in the drawing.

| | |
|---|---|
| OFF or 0 | Lineweights are not displayed |
| ON or 1 | Lineweights are displayed |

## LWUNITS

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether lineweight units are displayed in inches or millimeters.

| | |
|---|---|
| 0 | Inches |
| 1 | Millimeters |

# M System Variables

## MATSTATE

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Indicates whether the Materials window is open.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# MAXACTVP

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 64
Sets the maximum number of viewports that can be active at one time in a layout. MAXACTVP has no effect on the number of viewports that are plotted.

# MAXSORT

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1000
Sets the maximum number of symbol names or block names sorted by listing commands. If the total number of items exceeds this value, no items are sorted.

# MBUTTONPAN

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the behavior of the third button or wheel on the pointing device.

| | |
|---|---|
| 0 | Supports the action defined in the customization (CUI) file |
| 1 | Supports panning when you hold and drag the button or wheel |

# MEASUREINIT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** Varies by country/region
Controls whether a drawing you start from scratch uses imperial or metric default settings. Specifically, MEASUREINIT controls which hatch pattern and linetype files are used. The Drawing1.dwg that opens when you start the program is a drawing that is started from scratch.

| | |
|---|---|
| 0 | Imperial; uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings |
| 1 | Metric; uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings |

# MEASUREMENT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

| | |
|---|---|
| 0 | Imperial; uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings |
| 1 | Metric; uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings |

# MENUCTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the page switching of the screen menu.

| | |
|---|---|
| 0 | Screen menu does not switch pages in response to keyboard command entry |
| 1 | Screen menu does switch pages in response to keyboard command entry |

# MENUECHO

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Sets menu echo and prompt control bits. The value is the sum of the following:

| | |
|---|---|
| 1 | Suppresses echo of menu items (^P in a menu item toggles echoing) |
| 2 | Suppresses display of system prompts during menu |
| 4 | Disables ^P toggle of menu echoing |
| 8 | Displays input/output strings; debugging aid for DIESEL macros |

# MENUNAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"customization_file_name"
Stores the customization file name, including the path for the file name.

# MIRRTEXT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls how the MIRROR command reflects text.

| | |
|---|---|
| 0 | Retains text direction |
| 1 | Mirrors the text |

# MODEMACRO

## Quick Reference

**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Displays a text string on the status line, such as the name of the current drawing, time/date stamp, or special modes.

Use MODEMACRO to display a string of text, or use special text strings written in the DIESEL macro language to have the macro evaluated from time to time and base the status line on user-selected conditions.

# MSMSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores a value that indicates whether the Markup Set Manager is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# MSOLESCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**1.000
Controls the size of an OLE object with text that is pasted into model space. MSOLESCALE controls only the initial size. If the scale factor value is changed, existing OLE objects in the drawing are not affected.

A positive number scales by value. Zero (0) scales by the *DIMSCALE* value.

# MSLTSCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**1

Scales linetypes displayed on the model tab by the

| | |
|---|---|
| 0 | Linetypes displayed on the Model tab are not scaled by the annotation scale |
| 1 | Linetypes displayed on the Model tab are scaled by the annotation scale |

**NOTE** MSLTSCALE is set to 0 when you open drawings created in AutoCAD 2007 and earlier.

# MTEXTED

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**"Internal"
Sets the application for editing multiline text objects. You can specify a different text editor for the MTEXT command. If you set MTEXTED to internal or to null (.), the In-Place Text Editor is displayed. If you set MTEXTED to "OldEditor", the Multiline Text Editor is displayed. If you specify a path and the name of the executable file for another text editor or word processor, that path and file name instead are displayed instead.

If the multiline text object is fewer than 80 characters, you can enter **:lisped** to use the LISP editor.

Text editors other than the internal one show the formatting codes in paragraph text.

# MTEXTFIXED

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2

Sets the display size and position of multiline text.

When MTEXTED is set to “Internal,” multiline text is displayed in the In-Place Text Editor. When MTEXTED is set to “OldEditor,” multiline text is displayed in the Multiline Text Editor.

If MTEXTED is set to “Internal:”

| | |
|---|---|
| 0 or 1 | Displays the In-Place Text Editor and the text within it at the size, position, and rotation of the multiline text object in the drawing. |
| 2 | Displays the In-Place Text Editor and the text within it at the size, position, and rotation of the multiline text object in the drawing.<br>Text that would otherwise be difficult to read (if it is very small, very large, or is rotated) is displayed at a legible size and is oriented horizontally so that you can easily read and edit it. |

If MTEXTED is set to “OldEditor:”

| | |
|---|---|
| 0 | Displays the Multiline Text Editor and the text within it at the size and position of the multiline text object in the drawing. Text too large or too small to be edited is displayed at a minimum or maximum size. Text that is rotated is displayed horizontally. |
| 1 or 2 | Displays the Multiline Text Editor at a fixed position and size based on last use, and displays text in the editor at a fixed height. Text that is rotated is displayed horizontally. |

# MTJIGSTRING

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:** "abc"

Sets the content of the sample text displayed at the cursor location when the MTEXT command is started. The text string is displayed in the current text size and font. You can enter any string of up to ten letters or numbers or enter . (period) to display no sample text.

# MYDOCUMENTSPREFIX

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"pathname"
Stores the full path to the My Documents folder for the user currently logged on. These files are stored in the product folder under the Local Settings folder; for example, "c:\Documents and Settings\username\My Documents".

# N System Variables

# NOMUTT

### Quick Reference

**Type:** Short
**Saved in:** Not-saved
**Initial value:**0
Suppresses the message display (muttering) when it wouldn't normally be suppressed. Displaying messages is the normal mode, but message display is suppressed during scripts, AutoLISP routines, and so on.

| | |
|---|---|
| 0 | Resumes normal muttering behavior |
| 1 | Suppresses muttering indefinitely |

## NORTHDIRECTION

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Specifies the angle of the sun from north. This value is affected by the settings of the AUNITS and AUPREC system variables.

**NOTE** The angle is interpreted in the context of the world coordinate system (WCS). This value is completely separate from surveyor angular units, which are always set relative to the current UCS.

# O System Variables

## OBSCUREDCOLOR

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 257
Specifies the color of obscured lines. Value 0 designates byBlock, value 256 designates byLayer, and value 257 designates ByEntity. Values 1-255 designate an AutoCAD Color Index (ACI).

An obscured line is a hidden line made visible by changing its color and linetype. OBSCUREDCOLOR is available only in 2D views. In 3D views, the *VSOBSCUREDCOLOR* system variable is used.

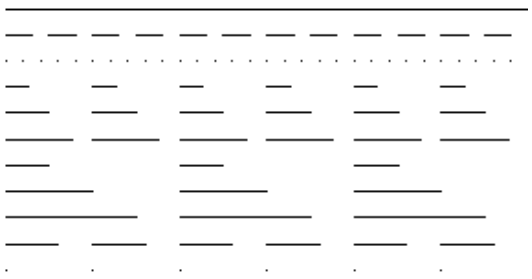
The OBSCUREDCOLOR setting is visible only if the *OBSCUREDLTYPE* system variable is turned on by setting it to a value other than 0.

# OBSCUREDLTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Specifies the linetype of obscured lines. Obscured linetypes are independent of zoom level, unlike regular linetypes. The linetype values are defined as follows:

| | |
|---|---|
| 0 | Off |
| 1 | Solid |
| 2 | Dashed |
| 3 | Dotted |
| 4 | Short Dash |
| 5 | Medium Dash |
| 6 | Long Dash |
| 7 | Double Short Dash |
| 8 | Double Medium Dash |
| 9 | Double Long Dash |
| 10 | Medium Long Dash |
| 11 | Sparse Dot |



An obscured line is a hidden line made visible by changing its color and linetype. OBSCUREDLTYPE is available only in 2D views. In 3D views, the *VSOBSCUREDLTYPE* system variable is used.

# OFFSETDIST

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:**-1.0000
Sets the default offset distance.

| | |
|---|---|
| <0 | Offsets an object through a specified point |
| 0 | Sets the default offset distance |

# OFFSETGAPTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0

Controls how potential gaps between segments are treated when closed polylines are offset.

| | |
|---|---|
| 0 | Fills the gaps by extending the polyline segments |
| 1 | Fills the gaps with filleted arc segments (the radius of each arc segment is equal to the offset distance) |
| 2 | Fills the gaps with chamfered line segments (the perpendicular distance to each chamfer is equal to the offset distance) |

# OLEFRAME

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**2
Controls whether a frame is displayed and plotted on all OLE objects in the drawing. The frame on an OLE object must be displayed in order for grips to be visible.

| | |
|---|---|
| 0 | Frame is not displayed and not plotted |
| 1 | Frame is displayed and is plotted |
| 2 | Frame is displayed but is not plotted |

# OLEHIDE

## Quick Reference

**Type:** Integer
**Saved in:** Registry

**Initial value:**0
Controls the display and plotting of OLE objects.

| | |
|---|---|
| 0 | All OLE objects are visible and plot |
| 1 | OLE objects are visible and plot in paper space only |
| 2 | OLE objects are visible and plot in model space only |
| 3 | No OLE objects are visible or plot |

# OLEQUALITY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**3
Sets the default plot quality for OLE objects. When OLEQUALITY is set to 3, the quality level is assigned automatically based on the type of object. For example, spreadsheets and tables are set to 0, color text and pie charts are set to 1, and photographs are set to 2.

| | |
|---|---|
| 0 | Monochrome |
| 1 | Low graphics |
| 2 | High graphics |
| 3 | Automatically Select |

# OLESTARTUP

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls whether the source application of an embedded OLE object loads when plotting. Loading the OLE source application may improve the plot quality.

| | |
|---|---|
| 0 | Does not load the OLE source application |
| 1 | Loads the OLE source application when plotting |

# OPMSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores a value that indicates whether the Properties palette is open, closed or hidden.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |
| 2 | Auto-hide: Open but only the title bar is displayed when the cursor moves off the palette |

# ORTHOMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Constrains cursor movement to the perpendicular. When ORTHOMODE is turned on, the cursor can move only horizontally or vertically relative to the UCS and the current grid rotation angle.

| | |
|---|---|
| 0 | Turns off Ortho mode |
| 1 | Turns on Ortho mode |

# OSMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**37
Sets running object snaps. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | NONe |
| 1 | ENDpoint |
| 2 | MIDpoint |
| 4 | CENter |
| 8 | NODe |
| 16 | QUAdrant |

| | |
|---|---|
| 32 | INTersection |
| 64 | INSertion |
| 128 | PERpendicular |
| 256 | TANgent |
| 512 | NEArest |
| 1024 | Clears all object snaps |
| 2048 | APParent Intersection |
| 4096 | EXTension |
| 8192 | PARallel |

To specify more than one object snap, enter the sum of their values. For example, entering 3 specifies the Endpoint (bitcode 1) and Midpoint (bitcode 2) object snaps. Entering 16383 specifies all object snaps.

When object snaps are switched off using the Osnap button on the status bar, a bitcode of 16384 (0x4000) is returned, in addition to the normal value of OSMODE. With this additional value, developers can distinguish this mode from Object Snap modes that have been turned off from within the Drafting Settings dialog box. Setting this bit toggles running object snaps off. Setting OSMODE to a value with this bit off toggles running object snaps on.

# OSNAPCOORD

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2

Controls whether coordinates entered on the command promptoverride running object snaps.

| | |
|---|---|
| 0 | Running object snap settings override keyboard coordinate entry |
| 1 | Keyboard entry overrides object snap settings |
| 2 | Keyboard entry overrides object snap settings except in scripts |

# OSNAPHATCH

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Obsolete. Use the OSOPTIONS system variable to control whether object snaps ignore hatch objects. The default setting, 0, improves performance.

| | |
|---|---|
| 0 | Object snaps ignore hatch objects |
| 1 | Object snaps treat hatch objects the same as other objects |

# OSNAPNODELEGACY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether the Node object snap can be used to snap to multiline text objects.

The number and location of nodes on a multiline text object snap depends on the vertical and horizontal justification of the multiline text object.

| | |
|---|---|
| 0 | Node object snap can be used with multiline text objects |
| 1 | Node object snap ignores multiline text objects |

# OSNAPZ

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls whether object snaps are automatically projected onto a plane parallel to the XY plane of the current UCS at the current elevation.

| | |
|---|---|
| 0 | Osnap uses the *Z*-value of the specified point |
| 1 | Osnap substitutes the *Z*-value of the specified point with the elevation (ELEV) set for the current UCS |

# OSOPTIONS

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:**3
Automatically suppresses object snaps on hatch objects and geometry with negative Z values when using a dynamic UCS.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Object snaps operate on hatch objects, and on geometry with negative Z values when using a dynamic UCS |

| | |
|---|---|
| 1 | Object snaps ignore hatch objects |
| 2 | Object snaps ignore geometry with negative Z values during use of a dynamic UCS |

# P System Variables

## PALETTEOPAQUE

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0

Controls whether windows can be made transparent. When transparency is unavailable or turned off, all palettes are opaque. Transparency is unavailable when palettes or windows are docked, when transparency is not supported by the current operating system, and when hardware accelerators are in use.

When transparency is available and turned on, you can use the Transparency option on the shortcut menu to set a different degree of transparency in individual palettes.

| | |
|---|---|
| 0 | Transparency turned on by user |
| 1 | Transparency turned off by user |
| 2 | Transparency unavailable though turned on by user |
| 3 | Transparency unavailable and turned off by user |

# PAPERUPDATE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls the display of a warning dialog when attempting to print a layout with a paper size different from the paper size specified by the default for the plotter configuration file.

| | |
|---|---|
| 0 | Displays a warning dialog box if the paper size specified in the layout is not supported by the plotter |
| 1 | Sets paper size to the configured paper size of the plotter configuration file |

# PDMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls how point objects are displayed. For information about values to enter, see the POINT (page 1095) command.

# PDSIZE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000

Sets the display size for point objects.

| | |
|---|---|
| 0 | Creates a point at 5 percent of the drawing area height |
| >0 | Specifies an absolute size |
| <0 | Specifies a percentage of the viewport size |

# PEDITACCEPT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Suppresses display of the Object Selected Is Not a Polyline prompt in PEDIT. The prompt is followed by "Do you want it to turn into one?" Entering y converts the selected object to a polyline. When the prompt is suppressed, the selected object is automatically converted to a polyline.

| | |
|---|---|
| 0 | The prompt is displayed |
| 1 | The prompt is suppressed |

# PELLIPSE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the ellipse type created with ELLIPSE.

| | |
|---|---|
| 0 | Creates a true ellipse object. |

| | |
|---|---|
| 1 | Creates a polyline representation of an ellipse |

# PERIMETER

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Not-saved
**Initial value:** 0.0000
Stores the last perimeter value computed by the AREA or LIST command.

Also stores perimeter values computed by *DBLIST*.

# PERSPECTIVE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** Varies
Specifies whether the current viewport displays a perspective view.

| | |
|---|---|
| 0 | Perspective view turned off |
| 1 | Perspective view turned on |

Perspective views are available only in model space, and only with 3D visual styles.

**NOTE** PERSPECTIVE is set to 0 when the drawing file or DXF file is saved to a file format earlier than AutoCAD 2007.

# PERSPECTIVECLIP

### Quick Reference

**Type:** Real
**Saved in:** User-settings
**Initial value:**5
Determines the location of eyepoint clipping. The value determines where the eye point clipping occurs as a percentage. Values can range between 0.01 and 10.0. If you select a small value, the z-values of objects will be compressed at the target view and beyond. If you select a value such as 0.5%, the clipping will appear very close to the eyepoint of the view. In some extreme cases it might be appropriate to use 0.1%, but it is recommended to change the setting to a higher value such as 5%.

# PFACEVMAX

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**4
Sets the maximum number of vertices per face.

# PICKADD

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether subsequent selections replace the current selection set or add to it.

| | |
|---|---|
| 0 | Turns off PICKADD. The objects and subobjects most recently selected become the selection set. Previously selec- |

| | |
|---|---|
| | ted objects and subobjects are removed from the selection set. Add more objects or subobjects to the selection set by pressing SHIFT while selecting. |
| 1 | Turns on PICKADD. Each object and subobject selected, either individually or by windowing, is added to the current selection set. To remove objects or subobjects from the set, press SHIFT while selecting. |

# PICKAUTO

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls automatic windowing at the Select Objects prompt.

| | |
|---|---|
| 0 | Turns off PICKAUTO. |
| 1 | Draws a selection window (for either a window or a crossing selection) automatically at the Select Objects prompt |

# PICKBOX

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 3
Sets the object selection target height, in pixels.

**NOTE** When PICKBOX is set to 0, selection previewing of objects is not available.

# PICKDRAG

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls the method of drawing a selection window.

| | |
|---|---|
| 0 | Draws the selection window using two points. Click the pointing device at one corner, and then click to select another corner. |
| 1 | Draws the selection window using dragging. Click one corner and drag the pointing device; release the button at the other corner. |

# PICKFIRST

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether you select objects before (noun-verb selection) or after you issue a command.

| | |
|---|---|
| 0 | Turns off PICKFIRST; you select objects after you issue a command |
| 1 | Turns on PICKFIRST; you select objects before you issue a command |

# PICKSTYLE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the use of group selection and associative hatch selection.

| | |
|---|---|
| 0 | No group selection or associative hatch selection |
| 1 | Group selection |
| 2 | Associative hatch selection |
| 3 | Group selection and associative hatch selection |

# PLATFORM

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:** Varies
Indicates which platform is in use.

# PLINEGEN

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0

Sets how linetype patterns generate around the vertices of a 2D polyline. Does not apply to polylines with tapered segments.

| | |
|---|---|
| 0 | Generates polylines to start and end with a dash at each vertex |
| 1 | Generates the linetype in a continuous pattern around the vertices of the polyline |

# PLINETYPE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2

Specifies whether optimized 2D polylines are used. PLINETYPE controls both the creation of new polylines with the PLINE command and the conversion of existing polylines in drawings from previous releases.

| | |
|---|---|
| 0 | Polylines in older drawings are not converted when opened; PLINE creates old-format polylines |
| 1 | Polylines in older drawings are not converted when opened; PLINE creates optimized polylines |
| 2 | Polylines in AutoCAD Release 14 or older drawings are converted when opened; PLINE creates optimized polylines |

For more information on the two formats, see the *CONVERT* command.

PLINETYPE also controls the polyline type created with the following commands: *BOUNDARY* (when object type is set to Polyline), *DONUT*, *PEDIT* (when selecting a line or arc), *POLYGON*, and *SKETCH* (when *SKPOLY* is set to 1).

# PLINEWID

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Stores the default polyline width.

# PLOTOFFSET

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls whether the plot offset is relative to the printable area or to the edge of the paper.

| | |
|---|---|
| 0 | Sets the plot offset relative to the printable area. |
| 1 | Sets the plot offset relative to the edge of the paper |

# PLOTROTMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2
Controls the orientation of plots.

| | |
|---|---|
| 0 | Rotates the effective plotting area so the corner with the Rotation icon aligns with the paper at the lower left for 0, top left for 90, top right for 180, and lower right for 270. |

| | |
|---|---|
| | *X* and *Y* origin offsets are calculated relative to the lower-left corner. |
| 1 | Aligns the lower-left corner of the effective plotting area with the lower-left corner of the paper. |
| 2 | Works the same as 0 value except that the *X* and *Y* origin offsets are calculated relative to the rotated origin position. |

# PLQUIET

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls the display of optional plot-related dialog boxes and nonfatal errors for scripts.

| | |
|---|---|
| 0 | Displays plot dialog boxes and nonfatal errors |
| 1 | Logs nonfatal errors and doesn't display plot-related dialog boxes |

# POLARADDANG

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**""
Stores additional angles for polar tracking and polar snap.

You can add up to 10 angles. Each angle can be up to 25 characters, separated with semicolons (;). The AUNITS system variable sets the format for display

of angles.Unlike POLARANG, POLARADDANG angles do not result in multiples of their values.

The bit value for the POLARMODE system variable must have 4 turned on for POLARADDANG to have an effect.

When using fractions of an angle, set the AUPREC system variable (angular precision) to a higher value. Otherwise, the POLARADDANG value will be rounded off.

# POLARANG

## Quick Reference

**Type:** Real
**Saved in:** Registry
**Initial value:**90
Sets the polar angle increment. Values are 90, 45, 30, 22.5, 18, 15,10, and 5.

# POLARDIST

## Quick Reference

**Type:** Real
**Saved in:** Registry
**Initial value:**0.0000
Sets the snap increment when the SNAPTYPE system variable is set to 1 (PolarSnap).

# POLARMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0

Controls settings for polar and object snap tracking. The setting is stored as a bitcode using the sum of the following values:

| Polar angle measurements | |
|---|---|
| 0 | Measure polar angles based on current UCS (absolute) |
| 1 | Measure polar angles from selected objects (relative) |

| Object snap tracking | |
|---|---|
| 0 | Track orthogonally only |
| 2 | Use polar tracking settings in object snap tracking |

| Use additional polar tracking angles | |
|---|---|
| 0 | No |
| 4 | Yes |

| Acquire object snap tracking points | |
|---|---|
| 0 | Acquire automatically |
| 8 | Press SHIFT to acquire |

**NOTE** In a 3D view, a tracking vector parallel to the *Z* axis of the UCS is also displayed, and the tooltip displays +Z and -Z for the angle depending on the direction along the *Z* axis.

# POLYSIDES

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved

**Initial value:**4
Sets the default number of sides for the POLYGON command. The range is 3 to 1024.

# POPUPS

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Displays the status of the currently configured display driver.

| | |
|---|---|
| 0 | Does not support dialog boxes, the menu bar, and icon menus |
| 1 | Supports these features |

# PREVIEWEFFECT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Specifies the visual effect used for previewing selection of objects.

| | |
|---|---|
| 0 | Dashed lines (the default display for selected objects) |
| 1 | Thickened lines |
| 2 | Dashed and thickened lines |

# PREVIEWFILTER

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:**7
Excludes specified object types from selection previewing. The setting is stored as a bitcode using the sum of the following values:

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Excludes nothing |
| 1 | Excludes objects on locked layers |
| 2 | Excludes objects in xrefs |
| 4 | Excludes tables |
| 8 | Excludes multiline text objects |
| 16 | Excludes hatch objects |
| 32 | Excludes objects in groups |

# PRODUCT

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**"AutoCAD"
Returns the product name.

# PROGRAM

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**"acad"
Returns the program name.

# PROJECTNAME

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**""
Assigns a project name to the current drawing. Used when an xref, image, or DWF underlay file is not found in its original path. The project name points to a section in the registry that can contain one or more search paths for each project name defined. Project names and their search directories are created from the Files tab of the Options dialog box.

Project names make it easier for users to manage xrefs, images, and DWF underlay when drawings are exchanged between customers, or if users have different drive mappings to the same location on a server.

If the xref, image, or DWF underlay file is not found at the original path, the project paths associated with the project name are searched. If the xref, image, or DWF underlay file is not found there, the AutoCAD search path is searched.

Assigns a project name to the current drawing. Used when an xref, image, or DWF underlay file is not found in its original path. The project name points to a section in the registry that can contain one or more search paths for each project name defined. Project names and their search directories are created from the Files tab of the Options dialog box.

# PROJMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Sets the current Projection mode for trimming or extending.

| | |
|---|---|
| 0 | True 3D mode (no projection) |
| 1 | Project to the *XY* plane of the current UCS |
| 2 | Project to the current view plane |

# PROXYGRAPHICS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Specifies whether images of proxy objects are saved in the drawing.

| | |
|---|---|
| 0 | Does not save image with the drawing; a bounding box is displayed instead |
| 1 | Saves image with the drawing |

# PROXYNOTICE

## Quick Reference

**Type:** Integer

**Saved in:** Registry
**Initial value:** 1
Displays a notice when a proxy is created. A proxy is created when you open a drawing containing custom objects created by an application that is not present. A proxy is also created when you issue a command that unloads a custom object's parent application.

| | |
|---|---|
| 0 | No proxy warning is displayed |
| 1 | Proxy warning is displayed |

# PROXYSHOW

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the display of proxy objects in a drawing.

| | |
|---|---|
| 0 | Proxy objects are not displayed |
| 1 | Graphic images are displayed for all proxy objects |
| 2 | Only the bounding box is displayed for all proxy objects |

# PROXYWEBSEARCH

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Specifies how the program checks for object enablers.

Object enablers allow you to display and use custom objects in drawings even when the ObjectARX application that created them is unavailable. PROXYWEBSEARCH is also controlled with the Live Enabler options on the System tab of the Options dialog box.

| | |
|---|---|
| 0 | Prevents checking for object enablers |
| 1 | Checks for object enablers only if a live Internet connection is present |

# PSLTSCALE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls linetype scaling of objects displayed in paperspace viewports.

| | |
|---|---|
| 0 | No special linetype scaling. Linetype dash lengths are based on the drawing units of the space (model or paper) in which the objects were created. Scaled by the global *LTSCALE* factor. |
| 1 | Viewport scaling governs linetype scaling. If *TILEMODE* is set to 0, dash lengths are based on paper space drawing units, even for objects in model space. In this mode, viewports can have varying magnifications, yet display linetypes identically. For a specific linetype, the dash lengths of a line in a viewport are the same as the dash lengths of a line in paper space. You can still control the dash lengths with LTSCALE |

When you change PSLTSCALE or use a command such as *ZOOM* with PSLTSCALE set to 1, objects in viewports are not automatically regenerated with the new linetype scale. Use the *REGEN* or *REGENALL* command to update the linetype scales in each viewport.

# PSOLHEIGHT

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 4 (imperial); 80 (metric)
Controls the default height for a swept solid object created with the POLYSOLID command. The value reflects the last entered height value when using the POLYSOLID command. You cannot enter 0 as the value.

# PSOLWIDTH

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.25 (imperial); 5 (metric)
Controls the default width for a swept solid object created with the POLYSOLID command. The value reflects the last entered width value when using the POLYSOLID command. You cannot enter 0 as the value.

# PSTYLEMODE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Indicates whether the current drawing is in a Color-Dependent or Named Plot Style mode.

| | |
|---|---|
| 0 | Uses named plot style tables in the current drawing |

| | |
|---|---|
| 1 | Uses color-dependent plot style tables in the current drawing |

# PSTYLEPOLICY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1

Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

| | |
|---|---|
| 0 | Drawing is set to use named plot styles. The plot style for new objects is set to the default defined in *DEFPLSTYLE*. The plot style for new layers is set to the default defined in *DEFLPLSTYLE*. |
| 1 | Drawing is set to use color-dependent plot styles. The plot style for an object is based on the object's color. |

# PSVPSCALE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0

Sets the view scale factor for all newly created viewports. The view scale factor is defined by comparing the ratio of units in paper space to the units in newly created model space viewports. The view scale factor you set is used with the VPORTS command. A value of 0 means the scale factor is Scaled to Fit. A scale must be a positive real value.

# PUBLISHALLSHEETS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls how the Publish dialog box list is populated when the PUBLISH command is issued.

| | |
|---|---|
| 0 | Only the current document's contents (layouts and/or model space) are automatically loaded in the publish list. |
| 1 | The contents (layouts and/or model space) of all open AutoCAD documents are automatically loaded in the publish list. |

# PUBLISHCOLLATE

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 1
Controls whether sheets are published as a single job

Publishing sheets as a single job requires all of the following:

- A plot or PDF driver that supports the multi-sheet plotting or printing option
- The page setup override option is selected, if publishing from the Sheet Set Manager.

| | |
|---|---|
| 0 | A published sheet set is processed one sheet at a time. Separate PLT and PDF files are created for each sheet. If the sheet set is published, the sheets might be interleaved with other plot jobs. |

| | |
|---|---|
| 1 | A published sheet set is processed as a single job. A multi-sheet PLT or PDF file is created. If the sheet set is published, it is never interleaved with other plot jobs. |

# PUCSBASE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:**""
Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only.

# Q System Variables

# QCSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**Varies
Determines whether the QuickCalc calculator is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# QTEXTMODE

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls how text is displayed.

| | |
|---|---|
| 0 | Turns off Quick Text mode; displays characters |
| 1 | Turns on Quick Text mode; displays a box in place of text |

# R System Variables

# RASTERDPI

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**300
Controls paper size and plot scaling when changing from dimensional to dimensionless output devices, or vice versa. Converts milimeters or inches to pixels, or vice versa. Accepts an integer between 100 and 32,767 as a valid value.

# RASTERPREVIEW

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1

Controls whether BMP preview images are saved with the drawing.

| | |
|---|---|
| 0 | No preview image is created |
| 1 | Preview image created |

# RECOVERYMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Controls whether drawing recovery information is recorded after a system failure.

| | |
|---|---|
| 0 | Recovery information is not recorded, the Drawing Recovery window does not display automatically after a system failure, and any recovery information in the system registry is removed |
| 1 | Recovery information is recorded, but the Drawing Recovery window does not display automatically after a system failure |
| 2 | Recovery information is recorded, and the Drawing Recovery window displays automatically in the next session after a system failure |

# REFEDITNAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved

**Initial value:**""
Displays the name of the reference being edited.

# REGENMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls automatic regeneration of the drawing.

| | |
|---|---|
| 0 | Turns off the *REGENAUTO* command |
| 1 | Turns on the *REGENAUTO* command |

# RE-INIT

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Reinitializes the digitizer, digitizer port, and acad.pgp file. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 1 | Digitizer input/output port reinitialization |
| 4 | Digitizer reinitialization |
| 16 | PGP file reinitialization (reload) |

To specify more than one reinitialization, enter the sum of the bitcode values. For example, enter 5 to specify both digitizer port (1) and digitizer reinitialization (4).

# REMEMBERFOLDERS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the default path displayed in standard file selection dialog boxes.

| | |
|---|---|
| 0 | Restores the behavior of AutoCAD 2000 and previous releases. When you start the program by double-clicking a shortcut icon, if a Start In path is specified in the icon properties, that path is used as the default for all standard file selection dialog boxes. |
| 1 | Uses standard Microsoft behavior. When you first start the program after installation, the default path in each standard file selection dialog box is *My Documents*. When you open or save a file to another folder, that folder is remembered for future file selection. The Start In folder specified for the shortcut icon is not used) |

# RENDERPREFSSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Stores a value that indicates whether the Render Settings palette is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# RENDERUSERLIGHTS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls whether user lights are translated during rendering

User lights are only rendered when this system variable is off. When on , user lights are rendered, even if default lighting is being used in the viewport.

| | |
|---|---|
| 0 | The current lights in viewport are used in the rendered scene, whether they are default lights or user lights. |
| 1 | User-lights are rendered even when default lighting is active. |

# REPORTERROR

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether an error report can be sent to Autodesk if the program closes unexpectedly.

Error reports help Autodesk diagnose problems with the software.

| | |
|---|---|
| 0 | The Error Report message is not displayed, and no report can be sent to Autodesk. |
| 1 | The Error Report message is displayed, and an error report can be sent to Autodesk |

If Windows XP is running, an error report can be sent to Microsoft if REPORTERROR is set to 0 and Windows is set to allow error reports.

# ROAMABLEROOTPREFIX

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"pathname"
Stores the full path to the root folder where roamable customizable files were installed. If you are working on a network that supports roaming, when you customize files that are in your roaming profile they are available to you regardless of which machine you are currently using.

These files are stored in the product folder under the Application Data folder; for example, "*c*:*Documents and Settings*\username*Application Data*\productname\version\language".

# RTDISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the display of raster images and OLE objects during Realtime ZOOM or PAN.

| | |
|---|---|
| 0 | Displays raster image and OLE content. |
| 1 | Displays outline only |

RTDISPLAY is saved in the current profile.

# S System Variables

## SAVEFIDELITY

### Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:**1
Controls whether the drawing is saved with visual fidelity

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Saved without visual fidelity |
| 1 | Saved with visual fidelity |

## SAVEFILE

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Registry
**Initial value:**"c:\Documents and Settings\username\Local Settings\TEMP\Drawing1.dwg"
Stores the current automatic save file name.

## SAVEFILEPATH

### Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**"c:\Documents and Settings\username\Local Settings\TEMP\"

Specifies the path to the directory for all automatic save files for the current session. You can also change the path on the Files tab in the Options dialog box.

# SAVENAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Displays the file name and directory path of the most recently saved drawing.

# SAVETIME

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**10
Sets the automatic save interval, in minutes.

| | |
|---|---|
| 0 | Turns off automatic saving. |
| >0 | Saves the drawing at intervals specified by the nonzero integer automatically |

The SAVETIME timer starts as soon as you make a change to a drawing. It is reset and restarted by a manual *QSAVE*, *SAVE*, or *SAVEAS*. The current drawing is saved to the path specified by the *SAVEFILEPATH* system variable. The file name is stored in the *SAVEFILE* system variable.

# SCREENBOXES

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores the number of boxes in the screen menu area of the drawing area. If the screen menu is turned off, SCREENBOXES is zero. On platforms that permit the drawing area to be resized or the screen menu to be reconfigured during an editing session, the value of this variable might change during the editing session.

# SCREENMODE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**3
Indicates the state of the display. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Text screen is displayed |
| 1 | Drawing area is displayed |
| 2 | Dual-screen display is configured |

# SCREENSIZE

## Quick Reference

(Read-only)

**Type:** 2D-point
**Saved in:** Not-saved
**Initial value:** Varies
Stores current viewport size in pixels (X and Y).

# SELECTIONANNODISPLAY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether alternate are temporarily displayed in a dimmed state when an object is selected

| | |
|---|---|
| 0 | Off |
| 1 | On |

The dimming intensity is controlled by the XFADECTL (page 1879) system variable.

# SELECTIONAREA

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the display of effects for selection areas. Selection areas are created by the Window, Crossing, WPolygon, and CPolygon options of SELECT.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# SELECTIONAREAOPACITY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**25
Controls the transparency of the selection area during window and crossing selection. The valid range is 0 to 100. The lower the setting, the more transparent the area. A value of 100 makes the area opaque. The SELECTIONAREA system variable must be on.

# SELECTIONPREVIEW

## Quick Reference

**Type:** Bitcode
**Saved in:** Registry
**Initial value:**3
Controls the display of selection previewing. Objects are highlighted when the pickbox cursor rolls over them. This selection previewing indicates that the object would be selected if you clicked. The setting is stored as a bitcode using the sum of the following values:

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Off |
| 1 | On when no commands are active |
| 2 | On when a command prompts for object selection |

# SETBYLAYERMODE

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:** 127
Controls which properties are selected for SETBYLAYER

The setting is stored as an integer using the sum of the following values:

| | |
|---|---|
| 0 | No properties are selected |
| 1 | Color property |
| 2 | Linetype property |
| 4 | Lineweight property |
| 8 | Material property |
| 16 | Plot Style property |
| 32 | Changes ByBlock to Bylayer |
| 64 | Includes blocks when changing ByBlock to ByLayer |

# SHADEDGE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 3

Controls the shading of edges in rendering.

| | |
|---|---|
| 0 | Faces shaded, edges not highlighted |
| 1 | Faces shaded, edges drawn in background color |
| 2 | Faces not filled, edges in object color |
| 3 | Faces in object color, edges in background color |

# SHADEDIF

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 70
Sets the ratio of diffuse reflective light to ambient light. The ratio is a percentage of diffuse reflective light when SHADEDGE is set to 0 or 1.

# SHADOWPLANELOCATION

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0
Controls the location of an invisible ground plane used to display shadows. The value is a location on the current Z axis. The ground plane is invisible, but it casts and receives shadows. Objects that are located below the ground plane are shadowed by it. The ground plane is used when the VSSHADOWS system variable is set to display either full shadows or ground shadows.

# SHORTCUTMENU

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 11

Controls whether Default, Edit, and Command mode shortcut menus are available in the drawing area. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Disables all Default, Edit, and Command mode shortcut menus, restoring AutoCAD Release 14 behavior. |
| 1 | Enables Default mode shortcut menus. |
| 2 | Enables Edit mode shortcut menus. |
| 4 | Enables Command mode shortcut menus whenever a command is active. |
| 8 | Enables Command mode shortcut menus only when command options are currently available at the command prompt. |
| 16 | Enables display of a shortcut menu when the right button on the pointing device is held down longer |

# SHOWHIST

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1

Controls the Show History property for solids in a drawing.

| | |
|---|---|
| 0 | Sets the Show History property to No (read-only) for all solids. Overrides the individual Show History property settings for solids. You cannot view the original objects that were used to create the solid. |
| 1 | Does not override the individual Show History property settings for solids. |
| 2 | Displays the history of all solids by overriding the individual Show History property settings for solids. You can view the original objects that were used to create the solid. |

# SHOWLAYERUSAGE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0

Displays icons in the Layer Properties Manager to indicate whether layers are in use. Setting this system variable to Off improves performance in the Layer Properties Manager.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# SHPNAME

## Quick Reference

**Type:** String
**Saved in:** Not-saved
**Initial value:** ""

Sets a default shape name that must conform to symbol- naming conventions. If no default is set, it returns "". Enter a period (.) to set no default.

# SIGWARN

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether a warning is presented when a file with an attached digital signature is opened. If the system variable is on and you open a file with a valid signature, the digital signature status is displayed. If the variable is off and you open a file, the digital signature status is displayed only if a signature is invalid. You can set the variable using the Display Digital Signature Information option on the Open and Save tab of the Options dialog box.

| | |
|---|---|
| 0 | Warning is not presented if a file has a valid signature |
| 1 | Warning is presented |

# SKETCHINC

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.1000
Sets the record increment for the SKETCH command.

# SKPOLY

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**0
Determines whether the SKETCH command generates lines or polylines.

| | |
|---|---|
| 0 | Generates lines |
| 1 | Generates polylines |

# SNAPANG

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0
Sets the snap and grid rotation angle for the current viewport. The angle you specify is relative to the current UCS.

When SNAPANG is set to a value other than 0, the lined grid will not display.

# SNAPBASE

## Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:**0.0000,0.0000
Sets the snap and grid origin point for the current viewport relative to the current UCS.

# SNAPISOPAIR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing

**Initial value:**0
Controls the isometric plane for the current viewport.

| | |
|---|---|
| 0 | Left |
| 1 | Top |
| 2 | Right |

# SNAPMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Turns the Snap mode on and off.

| | |
|---|---|
| 0 | Snap off |
| 1 | Snap on for the current viewport |

# SNAPSTYL

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the snap style for the current viewport.

| | |
|---|---|
| 0 | Standard (rectangular snap) |
| 1 | Isometric snap |

# SNAPTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Sets the type of snap for the current viewport.

| | |
|---|---|
| 0 | Grid, or standard snap. |
| 1 | PolarSnap. Snaps along polar angle increments. Use PolarSnap with polar and object snap tracking. |

# SNAPUNIT

## Quick Reference

**Type:** 2D-point
**Saved in:** Drawing
**Initial value:**0.5000,0.5000
Sets the snap spacing for the current viewport. If SNAPSTYL is set to 1, the X value of SNAPUNIT is adjusted automatically to accommodate the isometric snap.

Changes to this system variable are not reflected in the grid until the display is refreshed.

# SOLIDCHECK

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1

Turns the solid validation on and off for the current session.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# SOLIDHIST

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls the default History property setting for new and existing objects. When set to 1, composite solids retain a "history" of the original objects contained in the composite.

| | |
|---|---|
| 0 | Sets the History property to None for all solids. No history is retained. |
| 1 | Sets the History property to Record for all solids. All solids retain a history of their original objects. |

# SPLFRAME

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls the display of splines and spline-fit polylines.

| | |
|---|---|
| 0 | Does not display the control polygon for splines and spline-fit polylines. Displays the fit surface of a polygon |

| | |
|---|---|
| | mesh, not the defining mesh. Does not display the invisible edges of 3D faces or polyface meshes. |
| 1 | Displays the control polygon for splines and spline-fit polylines |

# SPLINESEGS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 8

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

Enter a non-zero integer between -32768 to 32767. If you set SPLINESEGS to a negative value, segments are generated using the absolute value of the setting and then a fit-type curve is applied to those segments. Fit-type curves use arcs as the approximating segments. Using arcs yields a smoother generated curve when few segments are specified, but the curve can take longer to generate.

# SPLINETYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 6

Sets the type of curve generated by the Spline option of the PEDIT command.

| | |
|---|---|
| 5 | Quadratic B-spline |
| 6 | Cubic B-spline |

# SSFOUND

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE must be set to 1 and the drawing file must be open for a successful search.

# SSLOCATE

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:**1
Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

| | |
|---|---|
| 0 | Does not open a drawing's sheet set with the drawing |
| 1 | Opens a drawing's sheet set with the drawing |

*SSMAUTOOPEN* and SSLOCATE must both be set to 1 to open a sheet set automatically in the Sheet Set Manager.

# SSMAUTOOPEN

## Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:**1

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

| | |
|---|---|
| 0 | Does not open the Sheet Set Manager automatically |
| 1 | Opens the Sheet Set Manager automatically |

SSMAUTOOPEN and *SSLOCATE* must both be set to 1 to open a sheet set automatically in the Sheet Set Manager.

# SSMPOLLTIME

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**60
Controls the time interval between automatic refreshes of the status data in a sheet set.

The SSMPOLLTIME timer sets the time in seconds between automatic refreshes of the status data of sheets in a sheet set. Valid values are 20-600. The *SSMSHEETSTATUS* system variable must be set to 2 for the timer to operate.

# SSMSHEETSTATUS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Controls how the status data in a sheet set is refreshed.

The status data for sheets in the current sheet set includes whether a sheet is locked and whether a sheet is missing (or found in an unexpected location). This status data can be updated automatically for all sheets.

To refresh the sheet set manually, use the Refresh Sheet Status button on the Sheet List tab of the Sheet Set Manager.

| | |
|---|---|
| 0 | Do not automatically refresh the status data in a sheet set |
| 1 | Refresh the status data when the sheet set is loaded or updated |
| 2 | Refresh the status data when the sheet set is loaded or updated, or at a time interval set by *SSMPOLLTIME* |

# SSMSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** Varies
Determines whether the Sheet Set Manager window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# STANDARDSVIOLATION

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 2
Specifies whether a user is notified of standards violations in the current drawing when a nonstandard object is created or modified.

Changes to the value of this system variable take effect only when Display Icons from Services is checked in the Tray Settings dialog box. To display the Tray Settings dialog box, click the down arrow at the right end of the tray on the status bar, and then click Tray Settings on the shortcut menu.

| | |
|---|---|
| 0 | Notification is turned off |
| 1 | An alert is displayed when a standards violation occurs in the drawing |
| 2 | An icon is displayed in the status bar tray when you open a file associated with a standards file and when you create or modify nonstandard objects |

# STARTUP

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0

Controls whether the Create New Drawing dialog box is displayed when a new drawing is started with NEW or QNEW. Also controls whether the Startup dialog box is displayed when the application is started.

If the FILEDIA system variable is set to 0, no dialog boxes are displayed, except for the Customize User Interface dialog box.

| | |
|---|---|
| 0 | Displays the Select Template dialog box, or uses a default drawing template file set in the Options dialog box, on the Files tab. |
| 1 | Displays the Startup and the Create New Drawing dialog boxes |

# STEPSIZE

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 6.0000
Specifies the size of each step when in walk or fly mode, in drawing units. You can enter any real number from 1E-6 to 1E+6.

# STEPSPERSEC

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 2
Specifies the number of steps taken per second when you are in walk or fly mode. You can enter any real number from 1 to 30.

# SUNPROPERTIESSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 0
Indicates whether the Sun Properties window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# SUNSTATUS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls whether the sun casts light in the current viewport.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# SURFTAB1

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**6
Sets the number of tabulations to be generated for the RULESURF and TABSURF commands. Also sets the mesh density in the M direction for the REVSURF and EDGESURF commands.

# SURFTAB2

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**6
Sets the mesh density in the N direction for the REVSURF and EDGESURF commands.

# SURFTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 6
Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.

| | |
|---|---|
| 5 | Quadratic B-spline surface |
| 6 | Cubic B-spline surface |
| 8 | Bezier surface |

# SURFU

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 6
Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects. Valid values are 0 through 200. Meshes are always created with a minimum surface density of 2.

# SURFV

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 6

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.. Valid values are 0 through 200. Meshes are always created with a minimum surface density of 2.

# SYSCODEPAGE

### Quick Reference

(Read-only)
**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Indicates the system code page, which is determined by the operating system. To change the code page, see Help in your operating system.

# T System Variables

# TABLEINDICATOR

### Quick Reference

**Type:** Integer
**Saved in:** User-settings
**Initial value:**1
Controls the display of row numbers and column letters when the In-Place Text Editor is open for editing a table cell.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# TABLETOOLBAR

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the display of the Table toolbar.

| | |
|---|---|
| 0 | The Table toolbar is never displayed. |
| 1 | The Table toolbar is displayed upon selection of a table cell. |

# TABMODE

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls the use of the tablet. For more information on using and configuring a tablet, see the *TABLET* command.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# TARGET

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing

**Initial value:**0.0000,0.0000,0.0000
Stores the location (as a UCS coordinate) of the target point for the current viewport.

# TBCUSTOMIZE

## Quick Reference

**Type:** Switch
**Saved in:** Registry
**Initial value:**1
Controls whether tool palette groups can be customized.

Controls whether the CUSTOMIZE command is enabled or disabled for tool palette group customization, and if the Customize option from the toolbar shortcut menu is displayed or not.

| | |
|---|---|
| 0 | Disables access to customization |
| 1 | Enables access to customization |

# TDCREATE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:**Varies
Stores the local time and date the drawing was created.

# TDINDWG

## Quick Reference

(Read-only)

**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Stores the total editing time, which is the total elapsed time between saves of the current drawing. The format is:

<number of days>.<decimal fraction of a day>

To compute the number of seconds, multiply the decimal fraction in TDINDWG by 86400 seconds.

# TDUCREATE

### Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Stores the universal time and date that the drawing was created.

# TDUPDATE

### Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Stores the local time and date of the last update/save.

# TDUSRTIMER

### Quick Reference

(Read-only)
**Type:** Real

**Saved in:** Drawing
**Initial value:** Varies
Stores the user-elapsed timer.

# TDUUPDATE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Stores the universal time and date of the last update/save.

# TEMPOVERRIDES

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Turns temporary override keys on and off. A temporary override key is a key that you can hold down to temporarily turn on or turn off one of the drawing aids that are set in the Drafting Settings dialog box; for example, Ortho mode, object snaps, or Polar mode.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# TEMPPREFIX

## Quick Reference

(Read-only)

**Type:** String
**Saved in:** Not-saved
**Initial value:**"c:\Documents and Settings\username\Local Settings\Temp\"
Contains the directory name (if any) configured for placement of temporary files, with a path separator appended.

# TEXTEVAL

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Controls how text strings entered with TEXT (using AutoLISP) or with -TEXT are evaluated.

| | |
|---|---|
| 0 | All responses to prompts for text strings and attribute values are taken literally. |
| 1 | All text starting from an opening parenthesis [(] or an exclamation mark [!] is evaluated as an AutoLISP expression, as for nontextual input. |

The *TEXT* command takes all input literally regardless of the setting of TEXTEVAL unless it is executed completely with a script or AutoLISP expression. The -TEXT command honors the setting of TEXTEVAL.

# TEXTFILL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls the filling of TrueType fonts while plotting and rendering.

| | |
|---|---|
| 0 | Displays text as outlines |

| | |
|---|---|
| 1 | Displays text as filled images |

# TEXTOUTPUTFILEFORMAT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Provides Unicode options for plot and text window log files.

| | |
|---|---|
| 0 | ANSI format |
| 1 | UTF-8 (Unicode) |
| 2 | UTF-16LE (Unicode) |
| 3 | UTF-16BE (Unicode) |

# TEXTQLTY

## Quick Reference

Sets the resolution tessellation fineness of text outlines

**Type:** Integer
**Saved in:** Not-saved
**Initial value:** 50

Sets the resolution tessellation fineness of text outlines for TrueType fonts while plotting and rendering. 0 represents no effort to refine the smoothness of the text; 100 represents a maximum effort to smooth text characters. Lower values decrease resolution and increase plotting speed. Higher values increase resolution and decrease plotting speed.

Sets the resolution of TrueType fonts while plotting. Use integer values from 0 to 100. Lower values decrease resolution and increase plotting speed. Higher values increase resolution and decrease plotting speed.

# TEXTSIZE

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.2000 (imperial); 2.500 (metric)
Sets the default height for new text objects drawn with the current text style. TEXTSIZE has no effect if the current text style has a fixed height.

# TEXTSTYLE

### Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** STANDARD
Sets the name of the current text style.

# THICKNESS

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0.0000
Sets the current 3D thickness.

# TILEMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Makes the Model tab or the last layout tab current.

| | |
|---|---|
| 0 | Makes the last active layout tab (paper space) active |
| 1 | Makes the Model tab active |

# TIMEZONE

## Quick Reference

**Type:** Enum
**Saved in:** Drawing
**Initial value:**-8000
Sets the time zone for the sun in the drawing. The values in the table are expressed as hours and minutes away from Greenwich Mean Time. The geographic location you set also sets the time zone. If the time zone is not accurate, you can correct it in the Geographic Location dialog box or set the TIMEZONE system variable.

| | |
|---|---|
| -12000 | International Date Line West |
| -11000 | Midway Island, Samoa |
| -10000 | Hawaii |
| -9000 | Alaska |
| -8000 | Pacific Time (US & Canada), Tijuana |

| | |
|---|---|
| -7000 | Arizona |
| -7000 | Chihuahua, La Paz, Mazatlan |
| -7000 | Mountain Time (US & Canada) |
| -7001 | Arizona |
| -7002 | Mazatlan |
| -6000 | Central America |
| -6001 | Central Time (US & Canada) |
| -6002 | Guadalajara, Mexico City, Monterrey |
| -6003 | Saskatchewan |
| -5000 | Eastern Time (US & Canada) |
| -5001 | Indiana (East) |
| -5002 | Bogota, Lima, Quito |
| -4000 | Atlantic Time (Canada) |
| -4001 | Caracas, La Paz |
| -4002 | Santiago |
| -3300 | Newfoundland |
| -3000 | Brasilia |
| -3001 | Buenos Aires, Georgetown |

| | |
|---|---|
| -3002 | Greenland |
| -2000 | Mid-Atlantic |
| -1000 | Azores |
| -1001 | Cape Verde Is. |
| 0 | Universal Coordinated Time |
| 1 | Greenwich Mean Time |
| 2 | Casablanca, Monrovia |
| +1000 | Amsterdam, Berlin, Bern, Rome, Stockholm |
| +1001 | Brussels, Madrid, Copenhagen, Paris |
| +1002 | Belgrade, Bratislava, Budapest, Ljubljana, Prague |
| +1003 | Sarajevo, Skopje, Warsaw, Zagreb |
| +1004 | West Central Africa |
| +2000 | Athens, Beirut, Istanbul, Minsk |
| +2001 | Bucharest |
| +2002 | Cairo |
| +2003 | Harare, Pretoria |
| +2004 | Helsinki, Kyiv, Sofia, Talinn, Vilnius |
| +2005 | Jerusalem |

| | |
|---|---|
| +3000 | Moscow, St. Petersburg, Volograd |
| +3001 | Kuwait, Riyadh |
| +3002 | Baghdad |
| +3003 | Nairobi |
| +3300 | Tehran |
| +4000 | Abu Dhabi, Muscat |
| +4001 | Baku, Tbilisi, Yerevan |
| +4300 | Kabul |
| +5000 | Ekaterinburg |
| +5001 | Islamabad, Karachi, Tashkent |
| +5300 | Chennai, Kolkata, Mumbai, New Delhi |
| +5450 | Kathmandu |
| +6000 | Almaty, Novosibirsk |
| +6001 | Astana, Dhaka |
| +6002 | Sri Jayawardenepura |
| +6300 | Rangoon |
| +7000 | Bangkok, Hanoi, Jakarta |
| +7001 | Krasnoyarsk |

| | |
|---|---|
| +8000 | Beijing, Chongqing, Hong Kong, Urumqi |
| +8001 | Kuala Lumpur, Singapore |
| +8002 | Taipei |
| +8003 | Irkutsk, Ulaan Bataar |
| +8004 | Perth |
| +9000 | Osaka, Sapporo, Tokyo |
| +9001 | Seoul |
| +9002 | Yakutsk |
| +9300 | Adelaide |
| +9301 | Darwin |
| +10000 | Canberra, Melbourne, Sydney |
| +10001 | Guam, Port Moresby |
| +10002 | Brisbane |
| +10003 | Hobart |
| +10004 | Vladivostok |
| +11000 | Magadan, Solomon Is., New Caledonia |
| +12000 | Auckland, Wellington |
| +12001 | Fiji, Kamchatka, Marshall Is. |

| | |
|---|---|
| +13000 | Nuku'alofa |

# TOOLTIPMERGE

## Quick Reference

**Type:** Switch
**Saved in:** User-settings
**Initial value:** 0
Combines drafting tooltips into a single tooltip. The appearance of the merged tooltip is controlled by the settings in the Tooltip Appearance dialog box.

| | |
|---|---|
| 0 | Off |
| 1 | On |

# TOOLTIPS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the display of tooltips on toolbars.

| | |
|---|---|
| 0 | Turns off the display of toolbar tooltips |
| 1 | Turns on the display of toolbar tooltips |

# TPSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**Varies
Determines whether the Tool Palettes window is open or closed.

| | |
|---|---|
| 0 | Closed |
| 1 | Open |

# TRACEWID

## Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0500
Sets the default trace width.

# TRACKPATH

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls the display of polar and object snap tracking alignment paths.

| | |
|---|---|
| 0 | Displays full-screen polar and object snap tracking paths |

| | |
|---|---|
| 1 | Displays full-screen polar tracking path; displays object snap tracking path only between the alignment point and the From point to the cursor location |
| 2 | Displays full-screen object snap tracking path; does not display polar tracking path |
| 3 | Does not display polar tracking path; displays object snap tracking path only between the alignment point and the From point to the cursor location |

# TRAYICONS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether a tray is displayed on the status bar.

| | |
|---|---|
| 0 | Does not display a tray |
| 1 | Displays a tray |

# TRAYNOTIFY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls whether service notifications are displayed in the status bar tray.

| | |
|---|---|
| 0 | Does not display notifications |

| | |
|---|---|
| 1 | Displays notifications |

# TRAYTIMEOUT

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 0
Controls the length of time (in seconds) that service notifications are displayed. Valid values are 0 to 10.

# TREEDEPTH

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 3020
Specifies the maximum depth, that is, the number of times the tree-structured spatial index can divide into branches.

| | |
|---|---|
| 0 | Suppresses the spatial index entirely, eliminating the performance improvements it provides in working with large drawings. This setting assures that objects are always processed in database order. |
| >0 | Turns on spatial indexing. An integer of up to five digits is valid. The first three digits refer to model space, and the remaining two digits refer to paper space. |
| <0 | Treats model space objects as 2D (*Z* coordinates are ignored), as is always the case with paper space objects. Such a setting is appropriate for 2D drawings and makes more efficient use of memory without loss of performance |

You cannot use TREEDEPTH transparently.

# TREEMAX

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 10000000

Limits memory consumption during drawing regeneration by limiting the number of nodes in the spatial index (oct-tree).

By imposing a fixed limit with TREEMAX, you can load drawings created on systems with more memory than your system and with a larger *TREEDEPTH* than your system can handle. These drawings, if left unchecked, have an oct-tree large enough to eventually consume more memory than is available to your computer. TREEMAX also provides a safeguard against experimentation with inappropriately high *TREEDEPTH* values.

The initial default for TREEMAX is 10000000 (10 million), a value high enough to effectively disable TREEMAX as a control for *TREEDEPTH*. The value to which you should set TREEMAX depends on your system's available RAM. You get about 15,000 oct-tree nodes per megabyte of RAM.

If you want an oct-tree to use up to, but no more than, 2 megabytes of RAM, set TREEMAX to 30000 (2 x 15,000). If the program runs out of memory allocating oct-tree nodes, restart, set TREEMAX to a smaller number, and try loading the drawing again.

The program might occasionally run into the limit you set with TREEMAX. Follow the resulting prompt instructions. Your ability to increase TREEMAX depends on your computer's available memory.

# TRIMMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1

Controls whether selected edges for chamfers and fillets are trimmed.

| | |
|---|---|
| 0 | Leaves selected edges intact |
| 1 | Trims selected edges to the endpoints of chamfer lines and fillet arcs |

# TSPACEFAC

## Quick Reference

**Type:** Real
**Saved in:** Not-saved
**Initial value:** 1.0
Controls the multiline text line-spacing distance measured as a factor of text height. Valid values are 0.25 to 4.0.

# TSPACETYPE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Controls the type of line spacing used in multiline text. At Least adjusts line spacing based on the tallest characters in a line. Exactly uses the specified line spacing, regardless of individual character sizes.

| | |
|---|---|
| 1 | At Least |
| 2 | Exactly |

# TSTACKALIGN

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls the vertical alignment of stacked text.

| | |
|---|---|
| 0 | Bottom aligned |
| 1 | Center aligned |
| 2 | Top aligned |

# TSTACKSIZE

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**70
Controls the percentage of stacked text fraction height relative to selected text's current height. Valid values are from 25 to 125.

# U System Variables

# UCSAXISANG

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**90

Stores the default angle when rotating the UCS around one of its axes using the X, Y, or Z option of the UCS command. Its value must be entered as an angle in degrees (valid values are: 5, 10, 15, 18, 22.5, 30, 45, 90, 180).

# UCSBASE

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** WORLD
Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings. Valid values include any named UCS.

# UCSDETECT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls whether dynamic UCS acquisition is active or not.

| | |
|---|---|
| 0 | Not active |
| 1 | Active |

# UCSFOLLOW

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0

Generates a plan view whenever you change from one UCS to another. The UCSFOLLOW setting is saved separately for each viewport. If UCSFOLLOW is on for a particular viewport, a plan view is generated in that viewport whenever you change coordinate systems.

Once the new UCS has been established, you can use *DVIEW*, *PLAN*, *VIEW*, or *VPOINT* to change the view of the drawing. It will change to a plan view again the next time you change coordinate systems.

| | |
|---|---|
| 0 | UCS does not affect the view |
| 1 | Any UCS change causes a change to the plan view of the new UCS in the current viewport |

The setting of UCSFOLLOW is maintained separately for paper space and model space and can be accessed in either, but the setting is ignored while in paper space (it is always treated as if set to 0). Although you can define a non-world UCS in paper space, the view remains in plan view to the world coordinate system.

# UCSICON

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**3
Displays the UCS icon for the current viewport or layout. This system variable has the same name as a command. Use the SETVAR command to access this system variable.

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | No icon is displayed |
| 1 | On; the icon is displayed in the lower-left corner of the current viewport or layout |
| 2 | Origin; if the icon is on, the icon is displayed at the UCS origin, if possible |

The setting of this system variable is viewport and layout specific.

# UCSNAME

## Quick Reference

(Read-only)
**Type:** String
**Saved in:** Drawing
Stores the name of the current coordinate system for the current viewport in the current space. Returns a null string if the current UCS is unnamed.

# UCSORG

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
Stores the origin point of the current coordinate system for the current viewport in the current space. This value is always stored as a world coordinate.

# UCSORTHO

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Determines whether the related orthographic UCS setting is restored automatically when an orthographic view is restored.

| | |
|---|---|
| 0 | Specifies that the UCS setting remains unchanged when an orthographic view is restored |

| | |
|---|---|
| 1 | Specifies that the related orthographic UCS setting is restored automatically when an orthographic view is restored |

# UCSVIEW

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1
Determines whether the current UCS is saved with a named view.

| | |
|---|---|
| 0 | Does not save current UCS with a named view |
| 1 | Saves current UCS whenever a named view is created |

# UCSVP

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport. The setting of this system variable is viewport specific.

| | |
|---|---|
| 0 | Unlocked; UCS reflects the UCS of the current viewport |
| 1 | Locked; UCS stored in viewport, and is independent of the UCS of the current viewport |

# UCSXDIR

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:**1.0000,0.0000,0.0000
Stores the X direction of the current UCS for the current viewport in the current space.

The setting of this system variable is viewport specific.

# UCSYDIR

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:**0.0000,1.0000,0.0000
Stores the Y direction of the current UCS for the current viewport in the current space.

The setting of this system variable is viewport specific.

# UNDOCTL

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**21

Indicates the state of the Auto, Control, and Group options of the UNDO command. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | UNDO is turned off |
| 1 | UNDO is turned on |
| 2 | Only one command can be undone |
| 4 | Auto is turned on |
| 8 | A group is currently active |
| 16 | Zoom and pan operations are grouped as a single action |

# UNDOMARKS

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores the number of marks placed in the UNDO control stream by the Mark option. The Mark and Back options are not available if a group is currently active.

# UNITMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0

Controls the display format for units. By default, the format for displaying measured values differs slightly from the format used for entering them. (You cannot include spaces when entering measured values.)

| | |
|---|---|
| 0 | Displays fractional, feet-and-inches, and surveyor's angles in "report" format using spaces as delimiters |
| 1 | Displays fractional, feet-and-inches, and surveyor's angles in "input" format without including spaces and, in some cases, substituting dashes for spaces |

# UPDATETHUMBNAIL

## Quick Reference

**Type:** Bitcode
**Saved in:** Drawing
**Initial value:**15
Controls updating of the thumbnail previews in the Sheet Set Manager. The setting is stored as a bitcode using the sum of the following values:

The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Does not update thumbnail previews for sheet views, model space views, or sheets |
| 1 | Updates model space view thumbnail previews |
| 2 | Updates sheet view thumbnail previews |
| 4 | Updates sheet thumbnail previews |
| 8 | Updates thumbnail previews when sheets or views are created, modified, or restored |
| 16 | Updates thumbnail previews when the drawing is saved |

# USERI1-5

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Provides storage and retrieval of integer values. There are five system variables: USERI1, USERI2, USERI3, USERI4, and USERI5.

# USERR1-5

### Quick Reference

**Type:** Real
**Saved in:** Drawing
**Initial value:**0.0000
Provides storage and retrieval of real numbers. There are five system variables: USERR1, USERR2, USERR3, USERR4, and USERR5.

# USERS1-5

### Quick Reference

**Type:** String
**Saved in:** Not-saved
**Initial value:**""
Provides storage and retrieval of text string data. There are five system variables: USERS1, USERS2, USERS3, USERS4, and USERS5.

# V System Variables

## VIEWCTR

### Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** Varies
Stores the center of view in the current viewport. Expressed as a UCS coordinate.

## VIEWDIR

### Quick Reference

(Read-only)
**Type:** 3D-vector
**Saved in:** Drawing
**Initial value:** None
Stores the viewing direction in the current viewport, expressed in UCS coordinates. This describes the camera point as a 3D offset from the target point.

## VIEWMODE

### Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0 in 2D templates, 1 in 3D templates

Stores the View mode for the current viewport. The setting is stored as a bitcode using the sum of the following values:

| | |
|---|---|
| 0 | Turned off. |
| 1 | Perspective view active. |
| 2 | Front clipping on |
| 4 | Back clipping on. |
| 8 | UCS Follow mode on. |
| 16 | Front clip not at eye. If on, the front clip distance (*FRONTZ*) determines the front clipping plane. If off, FRONTZ is ignored, and the front clipping plane is set to pass through the camera point (vectors behind the camera are not displayed). This flag is ignored if the front-clipping bit (2) is off. |

# VIEWSIZE

## Quick Reference

(Read-only)
**Type:** Real
**Saved in:** Drawing
**Initial value:** Varies
Stores the height of the view displayed in the current viewport, measured in drawing units.

# VIEWTWIST

## Quick Reference

(Read-only)

**Type:** Real
**Saved in:** Drawing
**Initial value:** 0
Stores the view rotation angle for the current viewport measured relative to the WCS.

# VISRETAIN

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls the properties of xref-dependent layers. Controls visibility, color, linetype, lineweight, and plot styles.

| | |
|---|---|
| 0 | The layer table, as stored in the reference drawing (xref), takes precedence. Changes made to xref-dependent layers in the current drawing are valid in the current session only and are not saved with the drawing. When the current drawing is reopened, the layer table is reloaded from the reference drawing and the current drawing reflects those settings. The layer settings affected are On, Off, Freeze, Thaw, Color, Ltype, LWeight, and PStyle (available only in named-plot style drawings). |
| 1 | Xref-dependent layer changes made in the current drawing take precedence. Layer settings are saved with the current drawing's layer table and persist from session to session. |

# VPLAYEROVERRIDES

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Drawing

**Initial value:**1
Indicates if there are any layers with viewport (VP) property overrides for the current layout viewport.

| | |
|---|---|
| 0 | Current viewport does not have any associated layer property overrides |
| 1 | Current viewport has associated layer properry overrides |

# VPLAYEROVERRIDESMODE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**1
Controls whether layer property overrides associated with layout viewports are displayed and plotted.

| | |
|---|---|
| 0 | Layer property overrides are not displayed in layout viewports or plotted |
| 1 | Layer properry overrides are displayed in layout viewports and plotted |

# VPMAXIMIZEDSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0

Stores a value that indicates whether the viewport is maximized. The maximized viewport state is canceled if you start the PLOT command.

| | |
|---|---|
| 0 | Not maximized |
| 1 | Maximized |

# VSBACKGROUNDS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls whether backgrounds are displayed in the visual style applied to the current viewport.

| | |
|---|---|
| 0 | Off |
| 1 | On |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSEDGECOLOR

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** BYENTITY
Sets the color of edges in the visual style in the current viewport.

Value 0 designates ByBlock, value 256 designates ByLayer, and value 257 designates ByEntity. Values 1-255 designate an AutoCAD Color Index (ACI) color. True Colors and Color Book colors are also available.

Valid values for True Colors are a string of integers each from 0 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

If you have a color book installed, you can specify any colors that are defined in the book.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSEDGEJITTER

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** -2
Controls the degree to which lines are made to appear as though sketched with a pencil. Turn off the jitter effect by preceding the setting with a minus sign.

| | |
|---|---|
| 1 | Low |
| 2 | Medium |
| 3 | High |

**NOTE** Plot styles are not available for objects with the Jitter edge modifier applied.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSEDGEOVERHANG

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**-6
Makes lines extend beyond their intersection, for a hand-drawn effect. The range is 1 to 100 pixels. Turn off the overhang effect by preceding the setting with a minus sign.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSEDGES

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Controls the types of edges that are displayed in the viewport.

| | |
|---|---|
| 0 | No edges are displayed |
| 1 | Isolines are displayed |
| 2 | Facet edges are displayed |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSEDGESMOOTH

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Specifies the angle at which crease edges are displayed. The range is 0 to 180.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSFACECOLORMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls how the color of faces is calculated.

| | |
|---|---|
| 0 | Normal: Does not apply a face color modifier |
| 1 | Monochrome: Displays all faces in the color that is specified in the *VSMONOCOLOR* system variable. |
| 2 | Tint: Uses the color that is specified in the *VSMONOCOLOR* system variable to shade all faces by changing the hue and saturation values of the color. |
| 3 | Desaturate: Softens the color by reducing its saturation component by 30 percent |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSFACEHIGHLIGHT

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**-30
Controls the display of specular highlights on faces without materials in the current viewport. The range is -100 to 100. The higher the number, the larger the highlight. Objects with materials attached ignore the setting of VSFACEHIGHLIGHT when VSMATERIALMODE is on.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSFACEOPACITY

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**-60
Controls the transparency of faces in the current viewport. The range is -100 to 100. At 100, the face is completely opaque. At 0, the face is completely transparent. Negative values set the transparency level but turn off the effect in the drawing.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSFACESTYLE

## Quick Reference

**Type:** Integer

**Saved in:** Drawing
**Initial value:**1
Controls how faces are displayed in the current viewport.

| | |
|---|---|
| 0 | No style applied |
| 1 | Real: as close as possible to how the face would appear in the real world |
| 2 | Gooch: uses cool and warm colors instead of dark and light to enhance the display of faces that might be shadowed and difficult to see in a realistic display |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSHALOGAP

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Sets the halo gap in the visual style applied to the current viewport. The range is 0 to 100.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSHIDEPRECISION

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved

**Initial value:**0
Controls the accuracy of hides and shades in the visual style applied to the current viewport.

| | |
|---|---|
| 0 | Single precision; uses less memory |
| 1 | Double precision; uses more memory |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSINTERSECTIONCOLOR

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**7
Specifies the color of intersection polylines in the visual style applied to the current viewport. The initial value is 7, which is a special value that inverts the color (black or white) based on the background color.

Value 0 designates ByBlock, value 256 designates ByLayer, and value 257 designates ByEntity. Values 1-255 designate an AutoCAD Color Index (ACI) color. True Colors and Color Book colors are also available.

**NOTE** *INTERSECTIONCOLOR* controls the color of intersection polylines when the visual style is set to 2D Wireframe.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSINTERSECTIONEDGES

## Quick Reference

**Type:** Switch
**Saved in:** Drawing
**Initial value:** 0
Controls the display of intersection edges in the visual style applied to the current viewport.

**NOTE** *INTERSECTIONDISPLAY* controls the color of intersection polylines when the visual style is set to 2D Wireframe.

| | |
|---|---|
| 0 | Off |
| 1 | On |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSINTERSECTIONLTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Sets the linetype for intersection lines in the visual style applied to the current viewport. The range is 1 to 11.

| | |
|---|---|
| 1 | Solid |
| 2 | Dashed |
| 3 | Dotted |

| | |
|---|---|
| 4 | Short Dash |
| 5 | Medium Dash |
| 6 | Long Dash |
| 7 | Double Short Dash |
| 8 | Double Medium Dash |
| 9 | Double Long Dash |
| 10 | Medium Long Dash |
| 11 | Sparse Dot |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSISOONTOP

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Displays isolines on top of shaded objects in the visual style applied to the current viewport.

| | |
|---|---|
| 0 | Off |
| 1 | On |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSLIGHTINGQUALITY

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Sets the lighting quality in the current viewport.

| | |
|---|---|
| 0 | Shows facets |
| 1 | Appears smooth |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSMATERIALMODE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the display of materials in the current viewport.

| | |
|---|---|
| 0 | No materials are displayed |
| 1 | Materials are displayed, textures are not displayed |
| 2 | Materials and textures are displayed |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSMAX

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** Varies
Stores the upper-right corner of the current viewport's virtual screen. Expressed as a UCS coordinate.

# VSMIN

## Quick Reference

(Read-only)
**Type:** 3D-point
**Saved in:** Drawing
**Initial value:** Varies
Stores the lower-left corner of the current viewport's virtual screen. Expressed as a UCS coordinate.

# VSMONOCOLOR

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** 255,255,255
Sets the color for monochrome and tint display of faces in the visual style applied to the current viewport. The initial value is white.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSOBSCUREDCOLOR

## Quick Reference

**Type:** String
**Saved in:** Drawing
**Initial value:** BYENTITY
Specifies the color of obscured (hidden) lines in the visual style applied to the current viewport.

Valid values include ByLayer (256), ByBlock (0), ByEntity (257), and any AutoCAD Color Index (ACI) color (an integer from 1 to 255).

You can also specify a true color or a color book color. Valid values for true colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB. The True Color setting is entered as follows:

RGB:000,000,000

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSOBSCUREDEDGES

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls whether obscured (hidden) edges are displayed.

| | |
|---|---|
| 0 | Off |
| 1 | On |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSOBSCUREDLTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Specifies the linetype of obscured (hidden) lines in the visual style applied to the current viewport. The range is 1 to 11.

| | |
|---|---|
| 1 | Solid |
| 2 | Dashed |
| 3 | Dotted |
| 4 | Short Dash |
| 5 | Medium Dash |
| 6 | Long Dash |
| 7 | Double Short Dash |
| 8 | Double Medium Dash |
| 9 | Double Long Dash |
| 10 | Medium Long Dash |
| 11 | Sparse Dot |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSSHADOWS

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0
Controls whether a visual style displays shadows.

| | |
|---|---|
| 0 | No shadows are displayed |
| 1 | Ground shadows only are displayed |
| 2 | Full shadows are displayed |

**NOTE** To display full shadows, hardware acceleration is required. When Geometry Acceleration is off, full shadows cannot be displayed. (To access these settings, enter **3dconfig** at the command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.)

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSSILHEDGES

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**0

Controls display of silhouette edges of solid objects in the visual style applied to the current viewport.

| | |
|---|---|
| 0 | Off |
| 1 | On |

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSSILHWIDTH

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**5
Specifies the width in pixels of silhouette edges in the current viewport. The range is 1 to 25.

**NOTE** Existing visual styles are not changed when you enter a new value for this system variable. Any new value entered for this system variable temporarily creates an unsaved new visual style.

# VSSTATE

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**0
Stores a value that indicates whether the Visual Styles window is open or closed.

# VTDURATION

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 750
Sets the duration of a smooth view transition, in milliseconds. The valid range is 0 to 5000.

# VTENABLE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 3
Controls when smooth view transitions are used. Smooth view transitions can be on or off for panning and zooming, for changes of view angle, or for scripts. The valid range is 0 to 7.

| Setting | For pan/zoom | For rotation | For scripts |
|---|---|---|---|
| 0 | Off | Off | Off |
| 1 | On | Off | Off |
| 2 | Off | On | Off |
| 3 | On | On | Off |
| 4 | Off | Off | On |
| 5 | On | Off | On |
| 6 | Off | On | On |

| Setting | For pan/zoom | For rotation | For scripts |
|---|---|---|---|
| 7 | On | On | On |

# VTFPS

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**7
Sets the minimum speed of a smooth view transition, in frames per second. When a smooth view transition cannot maintain this speed, an instant transition is used. The valid range is 1 to 30.

# W System Variables

# WHIPARC

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether the display of circles and arcs is smooth.

| | |
|---|---|
| 0 | Circles and arcs are not smooth, but rather are displayed as a series of vectors |
| 1 | Circles and arcs are smooth, displayed as true circles and arcs |

# WHIPTHREAD

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:** 1

Controls whether to use an additional processor to improve the speed of operations such as ZOOM that redraw or regenerate the drawing. WHIPTHREAD has no effect on single processor machines.

| | |
|---|---|
| 0 | No multithreaded processing; restricts regeneration and redraw processing to a single processor. This setting restores the behavior of AutoCAD 2000 and previous releases. |
| 1 | Regeneration multithreaded processing only; regeneration processing is distributed across two processors on a multiprocessor machine. |
| 2 | Redraw multithreaded processing only; redraw processing is distributed across two processors on a multiprocessor machine. |
| 3 | Regeneration and redraw multithreaded processing; regeneration and redraw processing is distributed across two processors on a multiprocessor machine. |

When multithreaded processing is used for redraw operations (value 2 or 3), the order of objects specified with the DRAWORDER command is not guaranteed to be preserved for display but is preserved for plotting.

# WINDOWAREACOLOR

## Quick Reference

**Type:** Integer
**Saved in:** Registry

**Initial value:**5 (blue)
Controls the color of the transparent selection area during window selection. The valid range is 1 to 255. SELECTIONAREA must be on.

# WMFBKGND

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**Off
Controls the background display when objects are inserted in Windows metafile (WMF) format. The objects may be inserted using any of the following methods:

- Output to a Windows metafile using *WMFOUT*
- Copied to the Clipboard and pasted as a Windows metafile
- Dragged as a Windows metafile

| | |
|---|---|
| Off | The background color is transparent. The foreground color depends on the setting of *WMFFOREGND*. |
| On | The background color is the same as the current background color in the drawing, whether in model space or in a layout. The foreground color remains unchanged. |

# WMFFOREGND

## Quick Reference

**Type:** Integer
**Saved in:** Not-saved
**Initial value:**Off

Controls the assignment of the foreground color when objects are inserted in Windows metafile (WMF) format. The objects may be inserted using any of the following methods:

- Output to a Windows metafile using *WMFOUT*
- Copied to the Clipboard and pasted as a Windows metafile
- Dragged as a Windows metafile

WMFFOREGND applies only when *WMFBKGND* is set to Off.

| | |
|---|---|
| Off | The foreground and background colors are swapped if necessary to ensure that the foreground color is darker than the background color |
| On | The foreground and background colors are swapped if necessary to ensure that the foreground color is lighter than the background color |

# WORLDUCS

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Indicates whether the UCS is the same as the WCS.

| | |
|---|---|
| 0 | UCS differs from the WCS |
| 1 | UCS matches the WCS |

# WORLDVIEW

## Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:**1
Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

| | |
|---|---|
| 0 | UCS remains unchanged |
| 1 | UCS changes to the WCS for the duration of the command; the command input is relative to the current UCS |

# WRITESTAT

## Quick Reference

(Read-only)
**Type:** Integer
**Saved in:** Not-saved
**Initial value:**1
Indicates whether a drawing file is read-only or can be written to. For developers who need to determine write status through AutoLISP.

| | |
|---|---|
| 0 | Can't write to the drawing |
| 1 | Can write to the drawing |

# WSCURRENT

## Quick Reference

**Type:** String

**Saved in:** Not-saved
**Initial value:** Name of default workspace
Returns the current workspace name in the command line interface and sets a workspace to current.

# X System Variables

## XCLIPFRAME

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 0
Controls the visibility of xref clipping boundaries.

| | |
|---|---|
| 0 | Not visible |
| 1 | Visible |

## XEDIT

### Quick Reference

**Type:** Integer
**Saved in:** Drawing
**Initial value:** 1
Controls whether the current drawing can be edited in-place when being referenced by another drawing.

| | |
|---|---|
| 0 | Can't use in-place reference editing |
| 1 | Can use in-place reference editing |

# XFADECTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**50
Controls the fading intensity percentage for references being edited in-place. Valid values are from 0 to 90

Also controls the fading intensity percentage of alternate object representations that display in a dimmed state, such as alternate scale representations of annotative objects.

*SELECTIONANNODISPLAY* controls whether or not alternate scale representations of annotative objects are displayed.

# XLOADCTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2
Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

| | |
|---|---|
| 0 | Turns off demand-loading; the entire drawing is loaded. |
| 1 | Turns on demand-loading. Referenced drawings are kept open and locked. |
| 2 | Turns on demand-loading. Copies of referenced drawings are opened and locked; referenced drawings are not locked |

When XLOADCTL is set to 2, a copy of each referenced drawing file is stored in the folder specified by the XLOADPATH system variable or the temporary files folder (set in the Options dialog box). Additionally, xrefs load faster when

you work across a network: the performance enhancement is most pronounced when you open drawings with many xrefs.

# XLOADPATH

## Quick Reference

**Type:** String
**Saved in:** Registry
**Initial value:**"pathname"
Creates a path for storing temporary copies of demand-loaded xref files. For more information, see XLOADCTL.

# XREFCTL

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls whether external reference log (XLG) files are created.

| | |
|---|---|
| 0 | Does not write log files |
| 1 | Writes log files |

# XREFNOTIFY

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**2

Controls the notification for updated or missing xrefs.

| | |
|---|---|
| 0 | Disables xref notification |
| 1 | Enables xref notification. Notifies you that xrefs are attached to the current drawing by displaying the xref icon in the lower-right corner of the application window (the notification area of the status bar tray). When you open a drawing, alerts you to missing xrefs by displaying the xref icon with a yellow alert symbol (!). |
| 2 | Enables xref notification and balloon messages. Displays the xref icon as in 1 above. Also displays balloon messages in the same area when xrefs are modified. The number of minutes between checking for modified xrefs is controlled by the system registry variable XNOTIFYTIME. |

# XREFTYPE

## Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Controls the default reference type when attaching or overlaying an external reference.

| | |
|---|---|
| 0 | Attachment is the default |
| 1 | Overlay is the default |

# Z System Variables

## ZOOMFACTOR

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**60
Controls how much the magnification changes when the mouse wheel moves forward or backward. Accepts an integer between 3 and 100 as a valid value. The higher the number, the more the change.

## ZOOMWHEEL

### Quick Reference

**Type:** Integer
**Saved in:** Registry
**Initial value:**0
Toggles the direction of transparent zoom operations when you scroll the middle mouse wheel.

| | |
|---|---|
| 0 | Moves wheel forward zooms in; moving wheel backwards zooms out. |
| 1 | Move wheel forward zooms out; moving wheel backwards zooms in. |

# Utilities

This section describes the AutoCAD® utilities. You run these utilities separately from AutoCAD.

## In this part

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# Utilities

# 28

## In this chapter

- Attach Digital Signatures
- Batch Standards Checker
- SLIDELIB

# Attach Digital Signatures

## Quick Reference

Attaches a digital signature to files

Start menu (Windows) ➤ Programs ➤ Autodesk ➤ AutoCAD ➤ Attach Digital Signatures
(DOS) AcSignApply.exe

The Attach Digital Signatures dialog (page 1886) box is displayed.

# Attach Digital Signatures Dialog Box

## Quick Reference

Selects the files for attachment of digital signatures.

### Files to Be Signed

Lists the names of the files to be signed, the folder in which each file resides, and the current status of the file.

### Add Files

Opens the Select File dialog box, where you select the files for digital signature.

### Search Folders

Opens the Search Folders dialog box, where you can search for files in the folder you specify.

### Remove

Removes selected files from the Files to Be Signed list.

### Clear List

Removes all files from the Files to Be Signed list.

### Number of Files to Sign

Displays the number of files that the Attach Digital Signatures program will attempt to sign.

### Select a Digital ID (Certificate)

Displays a list of digital IDs that you can use to sign files. Includes information about the organization or individual to whom the digital ID was issued, the digital ID vendor who issued the digital ID, and when the digital ID expires.

### Signature Information

Provides a list of time services you can use to add a time stamp to your digital signature, the status of the time server connection, and a Comments area (to include information relevant to the digital signature or to the files you are signing).

### Get Time Stamp From

Provides a list of time servers you can use to time stamp your digital signature and includes the connection status of the time server.

### Time Service Status

Displays the connection status (Successfully Contacted Time Server or Could Not Contact Time Server) of the time service.

### Comment

Provides a place to include information relevant to the digital signature or to the files you are signing.

### Sign Files

Attaches a digital signature to every file listed in this dialog box.

## Search Folders Dialog Box

### Quick Reference

Searches for folders that contain files you want to digitally sign.

#### Start in Folder

Enters the name of the folder where you want to begin the search for files you want to sign.

#### Include Subfolders

Searches subfolders of the folder in the Start in Folder box.

#### Browse

Opens the Browse for Folder dialog box, where you can choose a folder to search.

#### Search for Files Named

Searches for the file type you specify.

You can specify DWG, DWS, and DWT files, as well as EXE and ZIP files generated by eTransmit.

# Batch Standards Checker

### Quick Reference

Audits a set of drawings for standards violations

Start menu (Windows) ➤ Programs ➤ Autodesk ➤ AutoCAD ➤ Batch Standards Checker

The Batch Standards Checker Window (page 1889) is displayed.

# Batch Standards Checker Window

## Quick Reference

The Batch Standards Checker audits a series of drawings for standards violations and creates an XML-based summary report detailing all violations. To use the Batch Standards Checker, you must first create a standards check file that specifies the drawings to audit and the standards files used for the audit.

The Batch Standards Checker contains the following tabs:

- Drawings (page 1889)
- Standards (page 1890)
- Plug-ins (page 1891)
- Notes (page 1891)
- Progress (page 1892)

The Batch Standards Checker toolbar (page 1892) contains additional options.

### Drawings Tab (Batch Standards Checker)

Allows you to create a list of drawings to audit for standards violations.

**Drawings to Check** Lists the drawings to audit for standards violations. To add a drawing, click Add Drawing. To remove a drawing, click Remove Drawing. An exclamation mark precedes any drawings that the Batch Standards Checker can't locate. Drawings are audited in the order in which they are listed. To reorder drawings in the list, click Move Up or Move Down.

**Add Standards File** Opens a standard file selection dialog box, where you can locate and select a drawing to audit.

**Remove Drawing** Removes a drawing from the list displayed in Drawings to Check.

**Move Up** Moves the currently selected drawing up one position in the list displayed in Drawings to Check.

**Move Down** Moves a standards file down one position in the list displayed in Standards Used for Checking All Drawings.

**Description** Provides summary information about the drawing file currently selected in Drawings to Check.

**Check External References of Listed Drawings** Determines if external references (xrefs) are added to the Drawings to Check list. If checked, external references are added to the list as soon as you start a batch standards audit.

### Standards Tab (Batch Standards Checker)

Allows you to specify what standards files are used to verify named objects during the batch standards audit.

**Check Each Drawing Using Its Associated Standards Files** Specifies auditing each drawing using the standards files that are associated with it. If this option is selected, the remaining options on the tab are not available.

**Check All Drawings Using the Following Standards Files** Specifies ignoring the standards files that are associated with individual drawings and using the ones you select in Standards Used for Checking All Drawings instead.

**Standards Used for Checking All Drawings** Lists the standards files used to audit the drawings. To add a standards file, click Add Standards File. To remove a standards file, click Remove Standards File. If conflicts arise between multiple standards in this list (for example, if two standards files specify layers of the same name but with different properties), the standards file that is shown first in the list takes precedence. To change the position of a standards file in the list, select it and click Move Up or Move Down.

**Add Standards File** Adds a standards file to the list displayed in Standards Used for Checking All Drawings.

**Remove Standards File** Removes a standards file from the list displayed in Standards Used for Checking All Drawings.

**Move Up** Moves a standards file up one position in the list displayed in Standards Used for Checking All Drawings.

**Move Down** Moves a standards file down one position in the list displayed in Standards Used for Checking All Drawings.

**Description** Provides summary information about the standards file currently selected in the list.

### Plug-ins Tab (Batch Standards Checker)

Lists the standards plug-ins that are installed on the current system. A standards plug-in is installed for each of the named objects for which standards can be defined (layers, dimension styles, linetypes, and text styles). In the future, it is expected that third-party applications will be able to install additional plug-ins.

**Plug-ins Used When Checking Standards** Displays a list of all the standards plug-ins on the current system. You specify at least one plug-in to use when auditing a drawing by selecting from the list. The selected plug-ins are used when checking standards for the entire series of drawings.

**Description** Provides summary information about the plug-in currently selected in the list.

### Notes Tab (Batch Standards Checker)

Allows you to add notes to the XML report.

**Enter Notes to Include in Report** Allows you to enter additional notes for inclusion in the report.

### Progress Tab (Batch Standards Checker)

Provides summary information about the status of the current batch standards audit.

### Batch Standards Checker Toolbar

| | | |
|---|---|---|
|  | New | Creates a new standards check file with a *.chx* file name extension. Standards check files specify what drawings and standards files are used by the batch audit. |
|  | Open | Opens a standard file selection dialog box, where you can select a standards check file. |
|  | Save | Saves the current standards check file. |
|  | Save As | Opens a standard file save dialog box, where you can specify a name and location to save a standards check file. |
|  | Start Check | Begins a batch audit using the currently loaded standards check file. This button is available only when you have added drawings, associated a standards file, and selected at least one plug-in to use when checking for standards violations. |

| | | |
|---|---|---|
|  | Stop Check | Stops a previously started batch audit operation. This button is available only if a batch audit is currently in progress. |
|  | View Report | Displays an HTML report summarizing the results of the batch audit. This button is available when a batch audit report is available for viewing. The contents of the report are included with the standards check file. |
|  | Export Report | Exports an HTML report that can be distributed to other users. This button is available only when a batch audit has been completed and a report is available to export. This button also allows you to optimize existing XML reports that were created with a previous version of the CAD Standards feature, in order to improve report printing quality. |

# SLIDELIB

## Quick Reference

Compiles slide files listed in an ASCII file into a slide library file

You can construct slide library (SLB) files from slide (SLD) files by using the SLIDELIB utility program supplied in the main program folder. The following operating system command line syntax constructs a slide library:

**slidelib library [ < slidelist ]**

where

*library* specifies the slide library file (extension *.slb*) into which the slide files (extension *.sld*) are added. *slidelist* specifies a list of slide files. If you want to specify a file extension, it must be *.sld*.

SLIDELIB reads a list of slide file names. This list is normally supplied by redirecting a list of files (one per line in another file created using a text editor or a user-supplied utility program) to standard input.

The operating system commands shown below will create a *slidelist* file that can be used with SLIDELIB. All the slide files you want to compile into a slide library should be placed in a single directory. At a DOS prompt (version 5.0 or newer), enter the following:

**dir *.sld /b > mylist**

This creates the *mylist* file, which you can pass to SLIDELIB. You can also create the *mylist* file with a text editor by listing the slide file names (and paths, if necessary), such as *lobby*, *d:\slides\office*, and *\aec\slides\stairs*. The slide file name, but not the drive and directory information, is saved in the library file. Because only the file name is included, a library can contain slides with the same name from different directories, but only one of the slides can be accessed.

To generate the slide library *mlib* from *mylist*, enter the following:

**slidelib mlib < mylist**

This entry creates the file *mlib.slb*, which contains the names and definitions of the slides listed in *mylist*.

---

**WARNING** Do not delete your original slides. SLIDELIB has no provision for updating a slide library once it is created. If you want to add or delete a slide, update the *slidelist* file and re-create the library with SLIDELIB. All the original slides must be present in order to do this.

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